CONCEPT-II Version 12.0
User's Manual

Institut für
Theoretische Elektrotechnik

www.tet.tuhh.de

Dr.-Ing. Heinz-D. Brüns, Dipl.-Ing. Angela Freiberg

Copyright: Institut für Theoretische Elektrotechnik, 2014

TUHH
Technische Universität Hamburg-Harburg

- Nov. 2014 -

Manual of the Program System

1 General description 4
1.1 Supported platforms and the code capabilities 4
1.2 Possible excitations 4
1.3 Post processing 5
1.4 Installation 5
1.5 Covered frequency interval, limits 5
1.6 Discretization 6
1.6.1 Rules for discretizing structures 7
1.7 Three essential steps for solving a problem 12
1.8 Some file names 13
1.9 Performing a typical simulation 14
1.9.1 Overview 19
1.9.1.1 Icons of the tool bar 20
1.9.2 The simulation project tree 22
1.9.2.1 Handling of the simulation project tree 23
1.9.2.2 Dielectric bodies 24
1.9.2.3 Wires 28
1.9.2.4 Frequency sampling 32
1.9.2.5 Voltage generators 34
1.9.2.6 Ports 35
1.9.2.7 Plane wave field 37
1.9.2.8 Impressed wire current 39
1.9.2.9 Solvers 40
1.9.2.10 Precision of the numerical solution 41
1.9.3 Post processing tools 43
2 Example: Top-loaded wire antenna on a table structure 52
The handling of the GUI shall be demonstrated by a simple example. 52
2.1.1 The discretization of the surface part 54
2.1.2 Post processing 59
2.1.2.1 Representation of the current distribution 60
2.1.2.2 Representation of 2D electromagnetic field distributions 61
2.1.2.3 EM fields along straight paths or at arbitrary locations in space 63
2.1.2.4 Creating 2D radiation diagrams 66
2.1.2.5 Creating 3D radiation diagrams 68
2.1.2.6 Creating Smith diagrams 71
3 Important topics 71
3.1 Dielectric bodies, types of surfaces 71
3.1.1 Bodies of high conducting material 72
3.1.2 Field-coupling through apertures 73
3.1.3 Multi-dielectric block with symmetry 74
3.1.4 More examples 75
3.1.5 Wires in dielectric bodies, wire junctions 77
3.1.6 Layers on bodies 78
3.1.6.1 Example of a cylinder with a lossy wall of high relative permeability 80
3.2 N-ports, S-parameters 81
3.2.1 Files 83
3.2.2 Computation of S-parameters 85
3.2.3 EM fields when port excitations are modified 86

3.3 RCS 92
3.4 Computation of line integral of E and H 94
3.4.1 Example: Line integral to measure the surface current 96
3.5 Integrating the Poynting vector on a triangulated surface 98
3.5.1 Example: Loaded circular wire loop 99
3.6 Discretizing structures 101
3.6.1 Spheres, plates, cones, cylinders etc. 102
3.6.2 Treatment of wires 110
3.6.3 Generating triangular and quadrangular plates 112
3.6.4 Using the boundary tool 116
3.6.5 Effective discretization of wire contour models 119
3.6.6 Using the point option for discretizing 121
3.6.7 Joining surface parts with unwanted gaps 124
3.7 Importing and exporting 126
3.7.1 Converting STL data into surface patch format 126
3.7.2 Converting the wire format into the .geo format of gmsh 129
3.7.3 Using the CONCEPT-II patch format for mesh generation in gmsh 131
3.7.4 Visualizing field and current distributions by means of gmsh 133
3.8 Time domain system responses by an IFT 134
3.8.1 The necessary steps in short 134
3.8.2 Key points for performing an IFT 137
3.8.3 Frequency sampling for an IFT 146
3.8.4 Special aspects: Impressed currents and lightning currents 148
3.9 Using CONCEPT-II for parallel computation 151
3.9.1 Linux 151
3.9.2 Windows 152
3.10 Format of surface patch files and wire files 152
3.11 Arrow scaling, xtrans, ytrans 154
3.12 Backup of project directories 155
3.13 Known problems 155
4 Theoretical background (method of moments, EFIE, MFIE) 156

1 General description

CONCEPT-II is under development at the Institut für Theoretische Elektrotechnik Theory at the Technische Universität Hamburg-Harburg. It is a program package for the numerical computation of electromagnetic radiation and scattering problems in the frequency domain. The code is based on the Method of Moments (MoM) for solving the electric integral equation (EFIE) in the case of metal structures and a set of coupled integral equations in the case of (lossy) dielectric bodies, see Sec. 4 . It is designed for the computation of currents, voltages and fields, which result from the electromagnetic excitation of wire or surface structures. The application of the package is limited to linear materials which have to be homogeneous and isotropic in sections.

1.1 Supported platforms and the code capabilities

PC running under Linux and Windows.

The code capabilities include the treatment of:

- Lumped circuit elements at any location on wires and metallic surfaces
- Arbitrary metallic surfaces, which are simulated by the electric field integral equation (EFIE, single-layer current distribution)
- Surface impedance
- Thin lossy sheets on ideally conducting surfaces (not yet described in this manual, thin sheet approximation, TSA)
- Anisotropic layer material (CFC, not yet described in this manual, currently only for surface structures)
- Lossy dielectric bodies modeled by double-layers of electric currents
- Bodies covered by thin layers of finite conductivity to analyze shielding problems for example
- Structures in free space and over an ideally conducting ground plane
- Various magnetic symmetry planes
- Parallel LU decomposition
- Parallel MLFMA
- ACA

1.2 Possible excitations

- Voltage generators and power generators (ports) on wires, edge generators on metallic surfaces
- Impressed line current source (current generator)
- Plane-wave fields of arbitrary polarization and direction of incidence

1.3 Post processing

Once the matrix equation resulting from the application of the MoM has been set up and solved, the current distribution is known. Based on this current distribution system responses such as electromagnetic fields at arbitrary positions in space can be can be computed and displayed.

Most important features:

- 3D and 2D radiation diagrams
- Monostatic RCS
- Current distributions on surfaces and wires
- System responses (I,U,E,H) as a function of frequency
- Time domain system responses by an inverse Fourier transform (IFT)
- 2D and 1D field distributions
- Electromagnetic fields at arbitrary observation points...

For a successful application of the code the user should be familiar with the basic theory. A short outline is given in Section 4.

1.4 Installation

A document available on the *CONCEPT-II* download web page contains details about the installation of the package.

1.5 Covered frequency interval, limits

The conventional MoM provides a dense complex system matrix. This requires *O(N2)* to store the matrix elements and *O(N3)* to solve the equation system by LU decomposition for example, with *N* as the number of basis functions used to approximate the physical current distribution. The investigation of electrically large structures may become uneconomic using the LU decomposition, at first depending on the available hardware resources. The state-of-the-art technique for large hight frequency problems is the multilevel fast multipole algorithm (MLFMA) which is available as a solver.

Each common edge between two neighboring surface elements (patches, triangles or quadrangles) leads to one unknown in the case of the EFIE (metal, single-layer current distribution) or to two unknowns in the case of a dielectric body (double-layer current distribution), respectively. The total number of unknowns *N* depends on the electrical size and on the geometrical complexity (“overall grid”) of the structure under investigation.

An important rule of thumb is that 8....10 segments or patches per wavelength are necessary on electrically large structure parts, leading to an increasing number of unknowns at higher frequencies.

The core memory to hold the system matrix can be determined as follows:

$N \cdot N * 16 / 1024^3$ (LU decomposition)

Hence an important limit of the problem size to be treated is determined by the available core memory (RAM). Note that *CONCEPT-II* does not include an out-of-core solver anymore starting with version 12.0. The amount of in-core memory used by CONCEPT-II should stay smaller (70%,..,80%) than the memory being physically available . Additional memory is necessary, depending on the structure under investigation.

Examples:
Number of unknowns *N*=1000 (small problem) → 0.0149 Gbyte
Number of unknowns *N*=10.000 (medium sized problem) → Memory necessary to hold the system matrix: 1.49 GByte
Number of unknowns *N*=100.000 (large problem) → 149 GByte

The smallest frequency which is possible is in the range a 1,...,10 kHz depending on the geometrical size and the type of integral equation used. Metallic structures (EFIE) with a typical dimension in the range of 1 meter or larger can be computed even at lower frequencies.

Transient system responses can be gained by computing an appropriate number of samples in the frequency domain and performing a successive inverse Fourier transform (Sec. 3.8).

1.6 Discretization

CONCEPT-II includes powerful tools for creating surface patch meshes (discretization). Surface patch meshes may comprise planar triangles and/or (non-planar) quadrangles.
Nevertheless, for sophisticated meshing purposes external tools such as *gmsh* (http://geuz.org/gmsh) or *salome* (http://www.salome-platform.org) could be very helpful. *Salome* includes various open source licenses and *Gmsh* is distributed under the terms of the GPL. Both are able to read CAD data formats such as IGES or STEP and are able to convert the corresponding surfaces into discretized structures of STL format, which in turn can be read by *CONCEPT-II*. How to use *gmsh* in conjunction with *CONCEPT-II* is outlined in Sec. *3.7.*

Tip	Formats like IGES (.iges files) or STEP (.step files) can easily be turned into the grids by *gmsh* (.msh files) and from this into the *CONCEPT-II* surface patch data format, see Section 3.10.

For usage of the *CONCEPT-II* import/export tool click on **Misc** → **Import/export** → the corresponding dialog opens as depicted below.

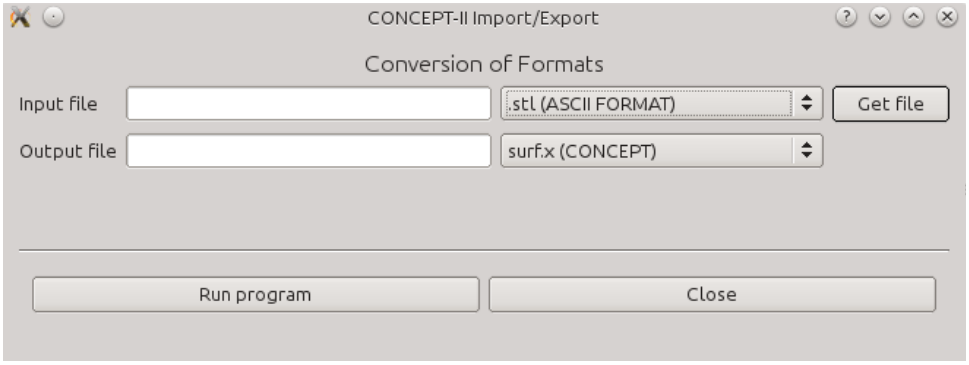


Fig 1.1: Tool for converting various files formats

The available features include:

- Conversion of STL data into the *CONCEPT-II* surface patch format.
- Conversion of the output of the *gmsh* tool into the *CONCEPT-II* surface patch data format.

Note: .msh files can be directly loaded into the simulation. The data is automatically converted into the *CONCEPT-II* surface patch format. The corresponding files have the suffix .surf .

1.6.1 Rules for discretizing structures

Important	A proper mesh is essential for the success and the convergence of the numerical computation.

It not sufficient that the structure under consideration is described 'somehow' by a surface patch pattern giving the right three-dimensional impression. Very important is a proper structure discretization related to the electromagnetic requirements. In this context we have to distinguish between electrically large and electrically small objects and structure parts, respectively.

Generally a wire is characterized by a straight line that extends between the start coordinates (Xstart, Ystart, Zstart) and the end coordinates (Xend, Yend, Zend). This is also the current reference direction. Between both ends a wire is subdivided into a number of segments. In the following it can be assumed that the number of wire segments corresponds to the number basis functions on a wire.

In general a surface is subdivided into a number of surface elements (“patches”). Triangles should be preferred.

- Wires and surfaces which are large in terms of the wavelength (electrically large) have to be discretized assuming 8,...,10 patches or wire segments per wavelength at least.
- The minimum number of basis functions should be three on electrically short wires, especially on wire grids. The minimum number of basis functions for short monopole antennas should be 6...8; 12...16 are sufficient on electrically small dipole antennas.
- No simple rule can be given for the discretization of electrically small structures. For parts which are small in terms of wavelength it may be important that geometrical details are modeled in a sufficient manner.

  Example: sphere with 1 m in diameter, f= 1 MHz → λ=300 m. Applying the high frequency rule we would have a typical patch size of 10 m which is not possible, of course. For a rough model of the sphere we have to apply 8,..,10 patches around the circumference to keep the outline of the sphere properly.

  In the low frequency region the user should answer the following question: "How many patches/wire elements are necessary to get a good approximation of the charge distribution under the assumption that only constant charge blocks are available on each triangle, quadrangle or wire segment".

  In general physical charge distributions are numerically approximated by a step functions. It is well known that we have a rapid increase of the surface charge distribution for example when approaching a sharp edge. Thus, if the electric field strength is of interest close to such an edge an appropriate grid refinement has to be introduced.
- For wires the well-known thin wire assumptions have to be fulfilled:

  $$L \gg a, \quad a \ll \lambda$$

  with L as the wire length, a as the radius and λ as the wavelength.

  Wires should be much longer than thick. Parallel wires should not get too close to each other. The radius of a wire should stay much smaller than the wavelength.

  In Fig. 1.2 can be seen how the characteristic impedance of a single wire line is behaving, when approaching the ground. The analytical solution refers to the red curve. The blue curves indicates the behavior of the characteristic impedance assuming a simple current filament on the wire axis. Starting at approximately r/h=0.3 (radius over height) the error gets more and more pronounced, even when assuming an ideal ground. One can imagine that an image wire is running at -h.

  Hence for an accurate solution the axial distance of two (or more) wires in parallel should not be closer than approximately 4 times the largest radius.

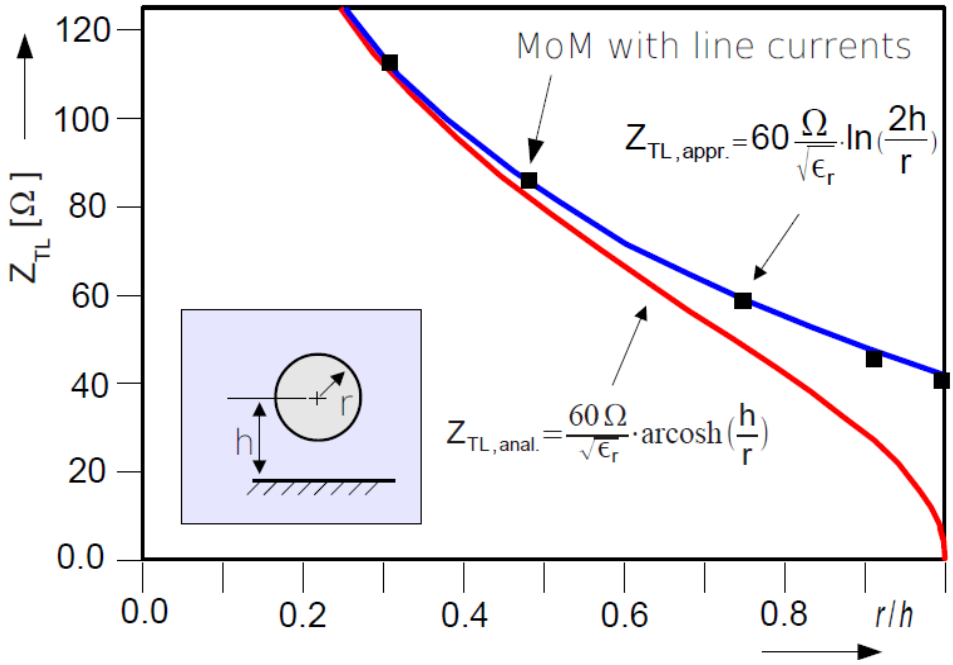


Fig 1.2: Characteristic impedance of a wire over a ground plane

- The ratio of the segment lengths of wires that are connected to each other should not exceed 10,...,15. Parallel wires should be discretized in the same way.
- Tiny patches should not be attached to much larger ones. Patches with a big ratio of edge lengths should be avoided, see Fig 1.3 and Fig 1.5 demonstrating bad examples. The same statement holds for triangles where 1 or 2 vertices are forming small angles.

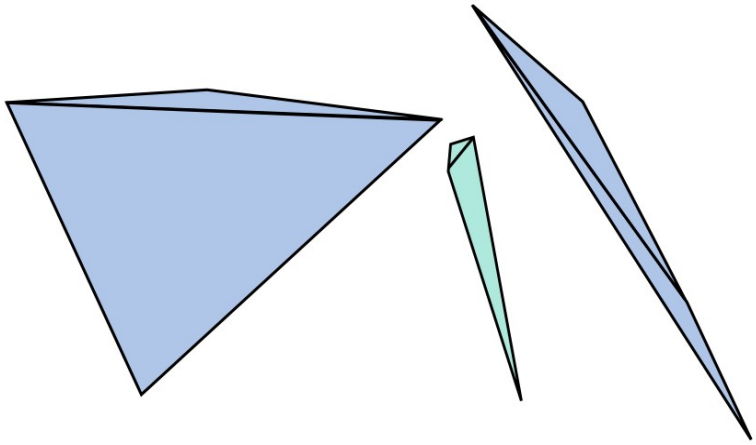


Fig 1.3: Examples of surface patch combinations that should be avoided

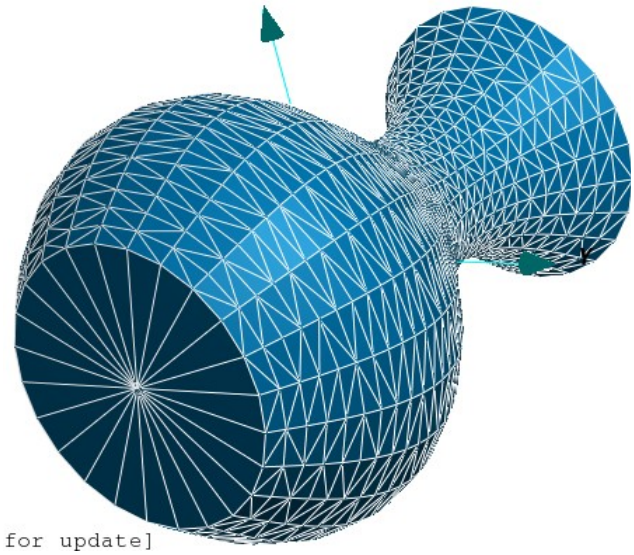


Fig 1.4: Example of a partly bad mesh. The discretization of this vase is good **except for the bottom**. Here we see only individual triangles extending between the rim and the center of the bottom disk - at least 5 to 6 were necessary (also for very low frequencies)! Here the highest frequency possible is determined by the long edges at the largest diameter of the vase. Hence the grid at the bottom is insufficient for both high and for low frequencies. From an electric point of view the bottom discretization makes absolutely no sense.

- Wires running parallel over a discretized ground plate, the typical size of the patches underneath the wire should be smaller then two times the height of the wire (perpendicular to the wire axis at least). Otherwise the characteristic impedance of the wire cannot be captured. Note that C' and L' are not input quantities – they are modeled by the sources on the local grid.In Fig 1.4 the bottom of the vase has been discretized according to pure visual aspects – this an error frequently encountered.

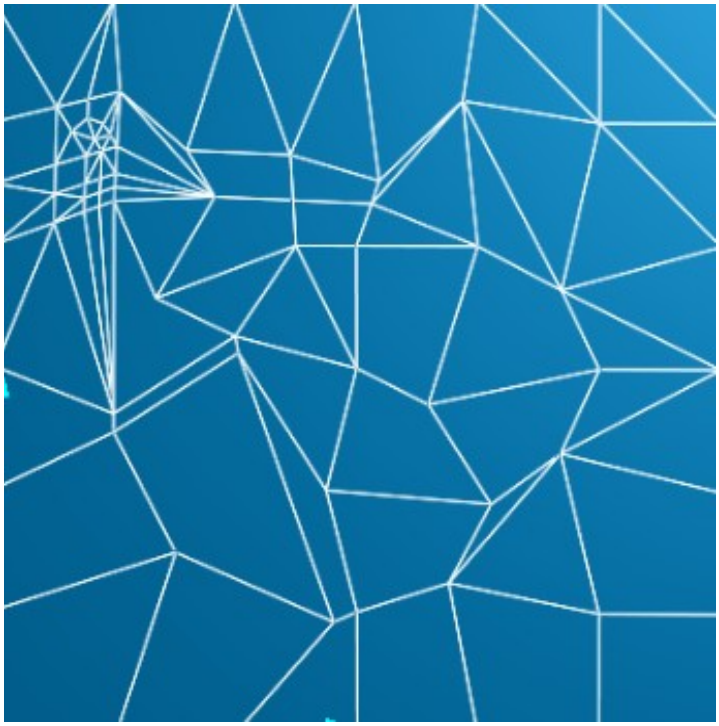


Fig 1.5: Example of a bad grid:

We see an **inhomogeneous mesh** which cannot be accepted including

- tiny triangles close to very large ones
- Triangles with the shape of needles (two corner close to 90 degrees or one corner close to 180 degrees)
- An erroneous quadrangle (corner angle larger then 180 degree), quadrangles with one corner angle close to 180 degrees or a quadrangle with one corner angle of only a few degrees.

Note	A mesh has to be generated always based on electromagnetic requirements not on visual aspects.

- Some simple examples: An electrically small disk with a wire antenna mounted in its center should have at least 5...6 patches between the center and the outer boundary. An isolated square plate should have at least 6 patches along each edge, even in case of a large wavelength.

Grid errors may be detected by a tool which can be found under
CAD tools → **CAD tools 2** → icon **Check surface structure** ., see Fig 1.8 ,

Project preview section.

It is essential that the grid is connected as otherwise “artificial slots” appear that will have a negative impact on the solution as surface current cannot flow over such locations. Fig 1.6 shows an example of a faulty grid. In practice such places are hard to detect by inspection of the structure in the *Display area.* Also in this case the mentioned tool might offer a significant service.

Fig 1.6: The center nodes of both grids are not in contact with each other. .This is a grid error as the current cannot pass such a line. Of course this is unsuitable method for modeling small existing slots! In such a case the patches close to the slot should not get larger then the slot width.

Once the solution of the problem has been computed, the question arises whether the results are convergent, numerically stable and reliable. Do the results confirm to the physical requirements? Checking the credibility of numerical results is not only strongly recommended, it is a must.

The most important techniques:

- The inspection of the power budget (input power = radiated power + losses)
- Physical credibility of results (is the solution at least basically sound)
- Model variation and parameter variation
- Check of the boundary conditions
- Convergence of computed quantities (change in patch size and segmentation within reasonable limits)
- Comparison with analytical solutions
- Comparison with measured data
- Comparison with the results other numerical methods

1.7 Three essential steps for solving a problem

The following steps are necessary to perform a numerical analysis:

1. Pre-processing
   Important and perhaps time-consuming in practical cases is the discretization of structures. The programs of the *CONCEPT-II* CAD tool box can be used, click on the **Cad tools** tab (see Fig 1.8, sub window *Project view*) After the surface patch structure has been discretized a number of additional input data has to be entered, such as wire coordinates, loads, kind of excitation, kind of symmetry, monitor probes, frequency steps etc., click the **Simulation** tab and fill the entries of the simulation *Project tree*.

2. Numerical computation
   This step involves setting up the equation system and solving it. Mainly depending on the number of unknowns N the computation can take a while. Setting up the system matrix is proportional to N^2 and solving the matrix equation by LU decomposition (standard solver) is proportional to N^3 . This means 8 times in solution time when doubling the number of unknowns.

3. Post processing
   Electromagnetic (EM) fields can be computed at arbitrary positions in space. Surface current distributions can be visualized, 2D or 3D radiation diagrams can be computed etc. (tab **Post processing**)

1.8 Some file names

The *CONCEPT-II* software package creates a number of ASCII and binary files in the current working directory. This following list gives a survey on the most important ones. It is advantageous for the user to know them.

concept.out
Created by the back-end. Contains results if the **Print** option (**Simulation** tab, simulation *Project tree*) is active. This file contains all current amplitudes, wire identifiers, number of unknowns, calculated system responses etc. *concept.out* is an ASCII file.

conerr.out
Contains error messages and warnings. Please carefully study in case of unexplainable program failure!

.in files
Almost all executables of the *CONCEPT-II* package require input data files. Examples: eh2d.in, plate.in. When existing, these files are automatically loaded by the GUI.

.out files
Many executables of the *CONCEPT-II* package produce result files in ASCII format. Examples: *concept.out*, *freqloop.out*, *eh1d.out*

surf.0 → The overall combined surface patch data file of the structure under investigation. If magnetic symmetry planes are existing only the surface parts that are in the symmetry plane need to be discretized. surf.0 contains the completed structure.
wire.0 → Contains the overall and combined wire structure, see remarks for *surf.0* .

Tip	It is not allowed to use the file names *surf.0* and *wire.0* when specifying a structure under the simulation *Project tree*

confreq.in → Contains the frequency steps

eh2d.movie → 2D EM field distribution, surface patch data format, data file *co_fmovie.asc*

co_cmovie.asc → current vectors to be displayed on *surf.0*

rad.n → Data of 3D radiation diagrams (surface patch data format)

co_ifl.asc → computed surface current distribution (to be displayed on *surf.0*)
co_ili.asc → computed wire current distribution
Erasing these two files in a project directory means that the whole perhaps time-consuming simulation has to be started again.

red-y-mat.asc → Reduced Y matrix/matrices in the case of multi port excitation.

freqi.asc, frequ.asc, freqzin.asc, freqe.asc, freqh.asc → Frequency domain system responses as a function of frequency, I(f), U(F), Z_{in}(f), E(f), H(f)

contour.asc → contour lines

1.9 Performing a typical simulation

A quick start.

Important for Windows users: Use only file names etc. without blanks!

Some useful and important key and mouse actions: **Help → Navigation**

It is recommended to start learning the handling of CONCEPT-II by means of the examples in the *$CONCEPT/demo* directory. We have 4 examples here and each

example is described step-by-step. The corresponding PDF files can be downloaded in addition to this manual and the installation instructions. A detailed description is also provided for the 'Table-antenna' example (CONCEPT/examples/Table-antenna), please refer to to Section 2.

A right-handed x,y,z coordinate system is assumed. In case of an ideal ground all structure parts have to be located in the upper space z> 0. The plane z=0 marks the ideal ground plane; surfaces and wires may be galvanically connected to it.

1. Start the *CONCEPT-II* graphical user interface

   This can be done by clicking the *CONCEPT-II* icon or by typing

   a) Linux: concept.sh <Enter> in the command line of a terminal

   b) Windows: concept.exe (Concept-II-12.0.bat in case of problems)

2. For a new simulation: click **File → New simulation**

   A dialogue opens with input fields for the *Project name*

   and the *Project directory.* Click **Browse** to choose an existing directory or to create a new directory.

   In case of a new structure open the **CAD tools** section by clicking the **CAD tools** tab (sub-window *Project View, see Fig 1.8*)

   Discretize the structure to be investigated (wires and/or surfaces), use the tools under tab **Cad tools 2** for this purpose**.** Simple bodies, cylinders, plates etc. can easily be discretized by clicking on one of these icons that appear in the Tool bar*:* .

   A structure which has already been loaded under **Simulation** and which shall be modified can easily be transferred into the CAD tools section: Press the **CAD tools** tab, right-click on the top tree entry: *Geometry from simulation → click on load simulation file(s).*

3. For an existing simulation: Click **File → Open simulation** to open the simulation. It is a good idea to start with the table antenna example which is in the $*CONCEPT/examples* directory.

   Input control files for *CONCEPT-II* end with the suffix .sim.

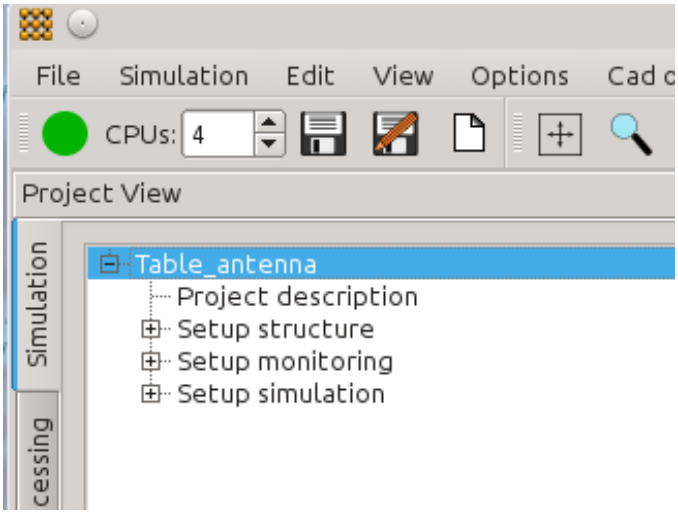
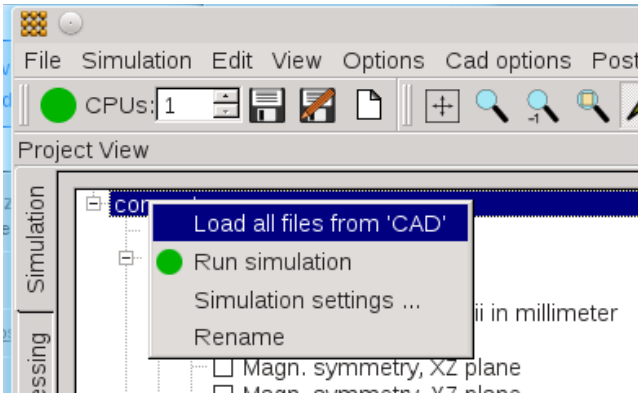


Fig 1.7: Example of the head section of the *CONCEPT-II* simulation project tree in 'Project View' section. Here we have the project **Table_antenna** from the input control file *Table_antenna.sim*

Important for users who are familiar with older versions of *CONCEPT-II*: you can select *concept.in* which was the former name of the input file. The contents of this file will be converted into the new file format which is completely different, but has ASCII format again.

The **Simulation** tab is now active.

(Newly created structures of the **Cad tools** section can be loaded into the **Simulation** by right clicking the root tree entry of the Project tree→ select **load all files from 'CAD'**, see figure below.)



4. Carefully check all items of the *p*roject tree. Right click opens a context

menu which is available in many cases.

Tip	Important for handling of the simulation project tree (section 'Project view') • Right click an item of the project tree. This opens a list box enabling the selection of dialog windows for example. • Double-click on an item (for example **Surfaces)** quits the sub-tree view. • Another double-click causes the sub-tree to become visible again Display area: Surfaces, nodes, wires, edges etc. are always selected by right click! Rotate mouse wheel: a possiblity to zoom in or out. Left click and drag: rotate the structure. Please notice **Help → Navigation**

Naming of wire files:

name_of_wire_file.wire or *wire.name_of_wire_file*

Naming of surface files:

name_of_surface_file.surf or *surf.name_of_surf_file*

Examples: *surf.1, box.surf, surf.box*

wire.1, grid.wire, wire.grid

Arbitrary names are possible in principle, however only file names according to the mentioned rules will be automatically recognized and listed.

5. Choose the number of CPUs (see spin box at the beginning of the tool bar, see Fig 1.8).
6. Start the simulation by clicking on the green **Run** icon on the left side of the tool bar, notice Section 1.5.

   The number of unknowns can be taken from the following action before starting the simulation: **Simulation** (M*enu bar*) → **Run simulation front-end** → The number of unknowns appears in the project status view section under the display area. A new simulation data file is written or an existing file will be overwritten; default name: *concept.sim*.

   See also tab **Log data** →**CONCEPT-II monitor**

7. After the simulation has finished start the post processing by clicking the **Post processing** tab. Output data of the back end run (contents of the ASCII file *concept.out*): tab **Log data → CONCEPT-II Simulation**

   Here a number of dialogs are available:

   - The current distribution on wires and patches can be displayed.

     Simple rule for a first validation: In general the vectors of the surface current distribution should follow smooth curves, they should look "beautiful". Jumps in current vectors from patch to patch are bad and cannot be accepted in large areas. The reason for "hot spots" or randomly distributed current vectors has to be investigated carefully. In many cases such phenomena are a hint for an insufficient grid (maybe too rough for a certain wavelength) or a non-convergent result in case of the MLFMA for example.

     16 phase intervals are a good choice for creating movies. Press **Animate** to start the movie.

     Right click into the *Scale widget* → a drop down list opens, select *Arrows,circles...* for scaling purposes (alter length of arrows, change colors, 3D arrows etc.)

   - 2D Electromagnetic field distributions can be computed

   - Radiation diagrams can be computed

   - ...

Note	Validation of the numerically gained results is a must-do!

8. Hint for users which are familiar with the *CONCEPT-II* versions prior to version 12.0:

   The whole package can still be operated at the prompt (terminal). This is recommended in case of an unexplainable behavior. Carefully study the output appearing on the screen.

   Start of the front end:
   concept.fe filename.sim <Enter>

Start of the back end:
concept.be <Enter> → Only a single CPU will be used

Linux:
mpirun -np no_of_processors concept.be <Enter> → The parallel back end runs with np processors.

Windows:
mpiexec -np no_of_processors concept.be.exe <Enter> → The parallel back end runs with np processors.

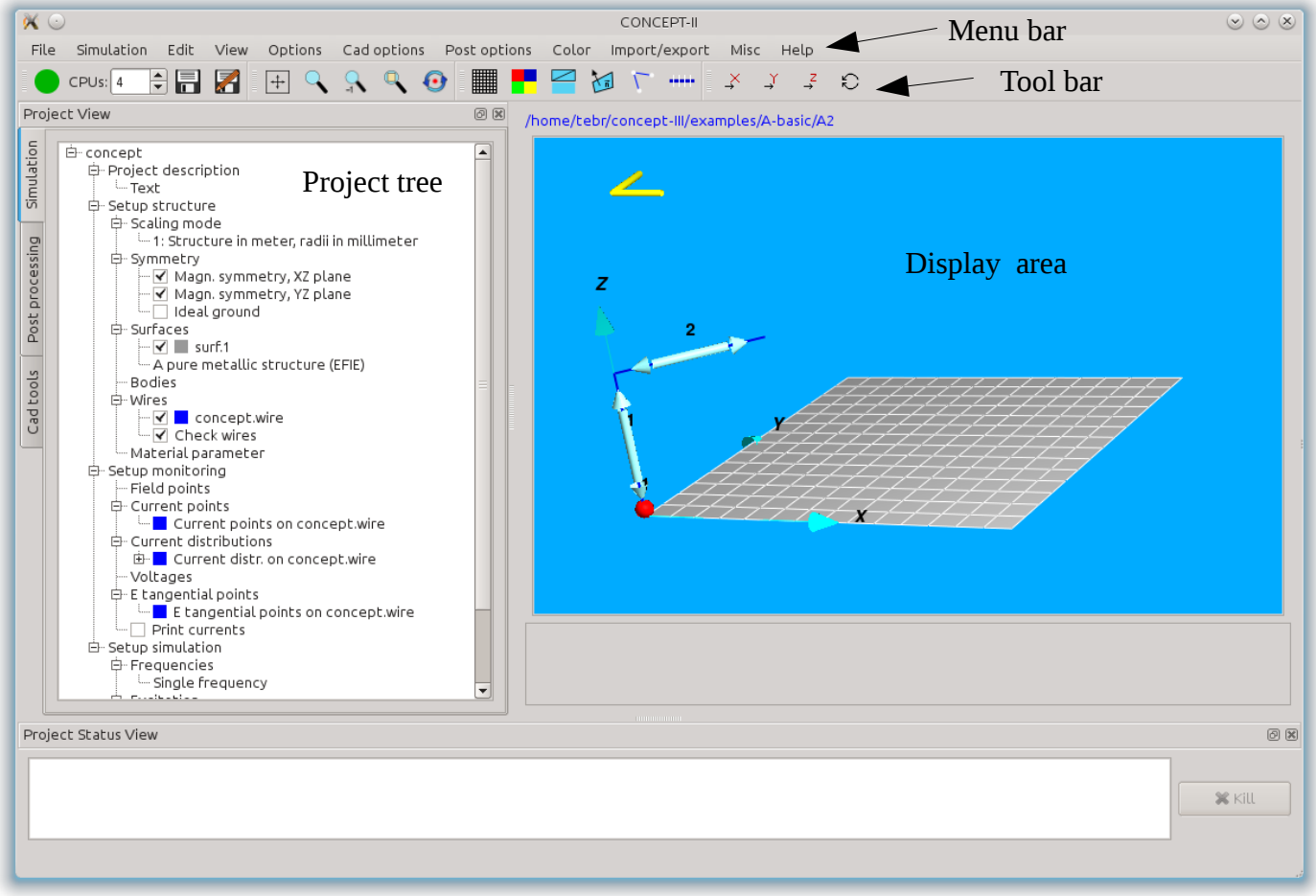


Fig 1.8: The *CONCEPT-II* main window

The *CONCEPT-II* main window is shown in Fig 1.8. In contrast to previous versions it has been designed for full screen usage.

1.9.1 Overview

- On the left side we have the 'Project View' section. The contents of this part

of the GUI changes depending on the active tab of the 'Project View' section . In Fig 1.8 the **Simulation** tab is being active, representing the simulation project tree. All input data for the numerical simulation (setting-up and solving the matrix equation(s)) has to be specified here.

- Display area
  - Left click, hold and drag causes the structure to rotate
  - Press keys <Ctrl>+<Shift>, left click, hold and drag → the selected sub-region will be enlarged.

    Alternative: activate , left click and drag the mouse pointer. If several steps of enlargement have been applied get back to the last step by clicking on . Show the initial representation by clicking on .
  - Press <Ctrl>,left click and drag → the structure is shifted in the *Display area* (the same action by clicking on , left click and drag)
  - Rotate the mouse wheel → structure is zooming in or out, depending on the direction of rotation; the zooming-in center is the current position of the mouse pointer.
  - All shortcuts: **Help → Navigation**
- In the top part we have the menu bar and the tool bar.

  The contents of the tool bar changes depending on tab being active in the 'Project View' section.
- At the bottom of the main window we have the project status view section where information of the currently running simulation is written into
- Directly underneath the display area we have a small section (a single line for text) where patch numbers, coordinates, physical quantities etc. are displayed on mouse over.

1.9.1.1 Icons of the tool bar

: If the **Simulation** tab is active the back end starts, setting up the equation system and solving it. When working under **CAD tools** pressing this icon may start a grid refinement or the generation of a surface or a similar action (the same action is initiated by typing on the <e> key).

: The dialogue window for curve representation opens; an example application is described in Sec. 2.1.2.3. Graphical output is possible in the formats png, svg and eps (CONCEPT-II gnuplot front end).

Indicates the x-y plane, sometimes helpful when added to the displayed structure.

: Tool for measuring distances. Select two different nodes by right clicks!

: Loaded surfaces parts appear in different colors

: Switch the grid on/off. Especially useful in case of fine large grids.

: Show normal vectors of all patches

: The end of wires are marked. Modify markers by **Cad options → Dot size (marker, wire ends)...**

: Show the segmentation of wires

: (“Picker” icon) By default information on patch numbers, node numbers, coordinates etc. is shown a small section underneath the display area on mouse over. In case of very large grids this might lead to an annoying delay when attempting to rotate the structure in the *Display area*. Disabling provides a smooth rotation again, of course depending on the performance the graphics card. It might be useful to deactivate this feature temporarily on a notebook or a computer with a less powerful graphics card.

: Enables the rotation of the structure around a selectable point . Right-click a node and drag the mouse. Known bug: after deactivating this feature the structure might appear somewhat deformed. Click on to get the original representation again.

: The end of wires is indicated by arrows. The reference direction is from the beginning to the end.

1.9.2 The simulation project tree

Data describing the structure under investigation need to be entered into the simulation project tree ('Project view'). Right clicking on an entry may open a drop-down list box where the user can choose from various items.

Note that a drop-down list is not available for all entries of the *p*roject tree. For example **Magn, symmetry, XZ plane** ('Project View'), *see Fig 1.8),* does not open a list.

The simulation tree consists of 4 major sections as demonstrated in Fig 1.9. Left click on the '+' box in front of an entries or a double click on the line opens the corresponding sub-tree.

The root entry is the project name to be specified when opening a new project directory, which is **Parallel-resistors** in the example according to Fig 1.9. The root entry is followed by these sections:

- **Project description**
- **Setup structure** →Wires and surfaces have to be specified here. Also: symmetry, types of boundary conditions and surface properties.
- **Setup monitoring** → Set probes for the computation of EM fields or probes for currents and/or voltages to be displayed as a function of frequency after the back end has finished.

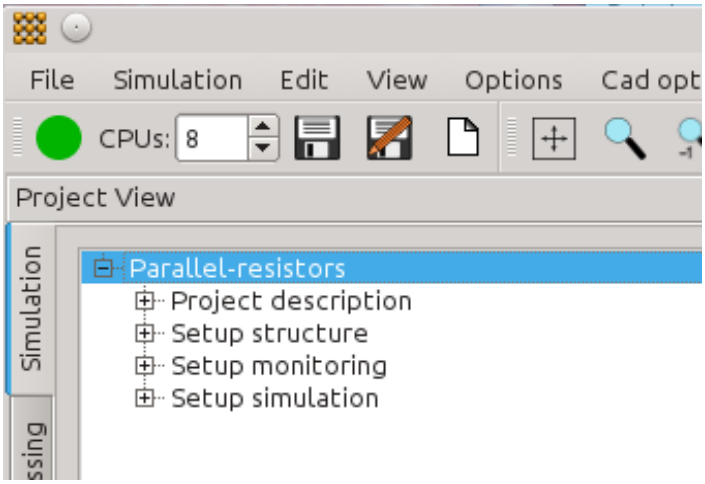


Fig 1.9: The sections of the **Simulation** *project tree*

- **Setup simulation** → The frequency sampling, the type of excitation and the type of solver have to be entered here.

1.9.2.1 Handling of the simulation project tree

As has been said already: Right-clicking on an entry of the project tree opens a drop-down list in most cases. Continue by selecting the required menu item.

Example:

A right click on **Surfaces** provides a drop-down list containing the entries

- **Load surface file**s
- **Load all surface(s) from CAD**
  - → All surface and wire files are loaded from the **Cad tools** section into the **Simulation** section
- **Unload all surface files**
- **A pure metallic structures**
  - → All dielectric interfaces are turned into metallic surfaces
- **Surface impedance**
  - → A conductivity and a thickness can be assigned a conductivity and a thickness can be assigned to metallic surfaces
- **Thin sheet approximation**
  - → An electrically thin and geometrically thin lossy sheet can be placed at one side of a PEC surface

Load all surface(s) from CAD → The corresponding file names appear as additional entries under **Surfaces;** an example can be seen in Fig 1.10 where a surface patch file named *surf.1* has been loaded.

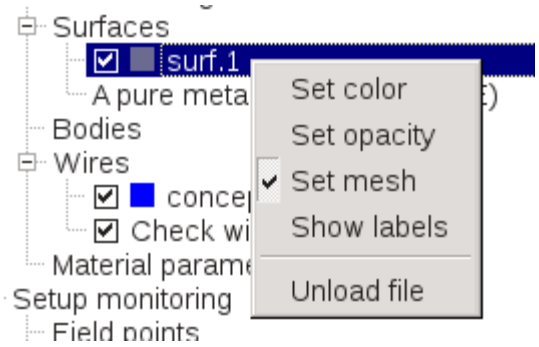


Fig 1.10: Drop-down list for modifying the properties of a surface patch file, here surf.1

Clicking on **Show labels** provides the possibility to display node numbers or patch numbers. It is recommended not to select all numbers for presentation in the case of large structures. If the numbers are too close to a surface patch file they can be

shifted into the direction of the patch normal vectors. An example of a simple metal structure can be found under *$CONCEPT/examples/Table_antenna*, see Sec. 2 for a detailed description. The **show labels** feature can be very helpful searching a patch by specifying its number only.

1.9.2.2 Dielectric bodies

- Bodies are finite regions in space with homogeneous material parameters, namely permittivity ϵ_r and conductivity σ. The user could imagine a lossy dielectric sphere as a simple example of a body, for details concerning theory refer to Section 4.
- On the surface of a pure dielectric body the following boundary conditions are matched:

  $$E_{t,internal} = E_{t,external}$$
  $$H_{t,internal} = H_{t,external} \qquad (1)$$

  Parts of a dielectric body may be covered by a metallic surface. In this case the boundary condition $E_{\tan} = 0$ is separately matched on both sides of this surface. Therefore surfaces that are bounding a body lead to double the number of unknowns compared to simple metallic surfaces in free space. Bodies are completely enclosed by two different electric current distributions in *CONCPT-II*. One current distribution is “responsible” for the internal EM field and the other one makes the external EM field. Both can be visualized.
- Surfaces that are bounding a dielectric region (a body) have to be completely closed.

  Surfaces are not possible in a symmetry plane; the user can think of the structure being completed on the other side of the symmetry plane, again yielding a completely closed overall surface.
- Wire can be placed in bodies. If a wire is penetrating a surface of a body, this wire has to be split into two parts: both sections must be attached to the same surface patch node, i.e. the wires sections must end at the same patch node. All patches sharing this node will be metalized automatically.
- Structure parts (wires or surfaces) inside dielectric bodies have to be discretized according to the internal wavelength.
- Metal cavities with a small aperture in air and filled with air can be considered as bodies: the apertures are modeled by a surface were the boundary condition (1) is matched. On the cavity wall which is described by another boundary $E_{\tan} = 0$ is matched separately at the internal and external side. Such a proceeding may be advantageous for a careful analysis of coupled-in EM fields. Note: Defining a completely closed metallic cavity as a body makes no sense.

- Surfaces where the boundary condition (1) is specified can be layered by thin lossy dielectric or lossy magnetic sheets. The thickness of such sheets should not be larger then 5...6 times the penetration depths resulting from the layer material.
- The typical dimension of bodies that can be computed depends on the internal conductivity – if it is too high the solution may become unstable.
- Rule: All patches of a (sub)surface bounding a body should have the same two regions on both sides.

  More details concerning the modeling of bodies are given in Secs. 3.1 and 4.

  Examples:

  $CONCEPT/examples/Three_cubes,
  $CONCEPT/examples/Parallel_resistors,
  $CONCEPT/examples/Bodies_in_bodies

The procedure to specify bodies is as follows, see Figs. 1.12 1.13 1.15 and shall be explained by an example.

As can be seen in Fig 1.11 we have two quarters of a cylinder attached to each other. Due to symmetry the problem comprises a full cylinder on an ideal ground plane with two bodies separated by an internal surface. Each cylindrical section may include a homogeneous (lossy) dielectric material.

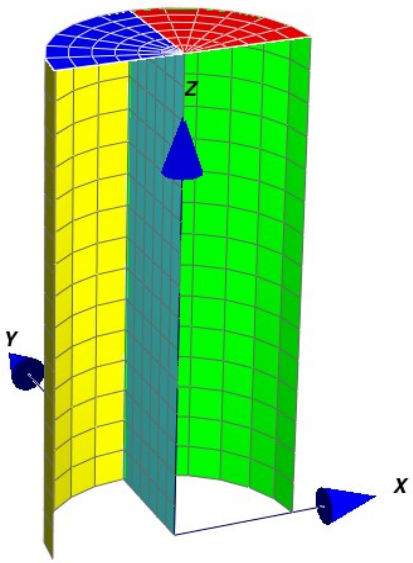


Fig 1.11: Example of two bodies attaches to each other on ideal (x-y plane). The assumed magnetic symmetry plane is the x-z plane. We have 5 surfaces bounding the bodies. The top and blue surface are assumed to be PEC. Assuming a certain conductivity in the bodies we can model a parallel resistor (example from $CONCEPT/examples/Parallel_resistors)

1. Load the surfaces describing the structure: **Surfaces → Load surface file(s).** In our example we have 5 surfaces
2. Right click **Bodies → Add bodies** → a new entry **body 1** appears
3. Right click **body 1** → the drop-down list shown in Fig 1.13 appears. Now the surfaces covering **body 1** can be selected either by mouse over (right-click on the corresponding surface) or by editing a list that appears when choosing **Surface selection by edit**. Continue in the same way defining **body 2**

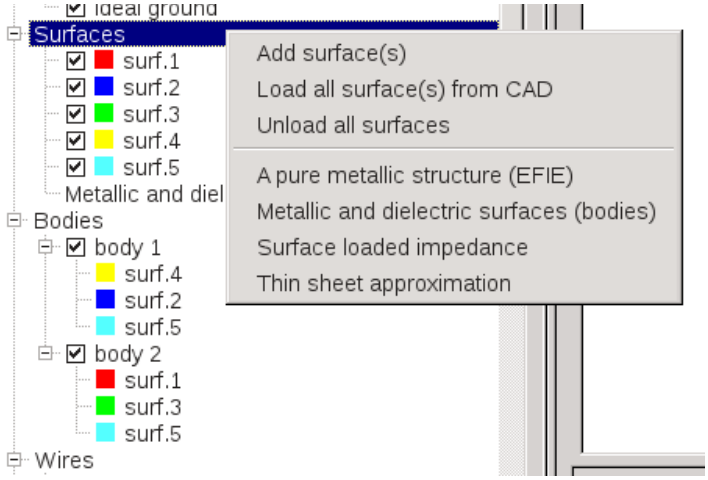


Fig 1.12: Surfaces forming two bodies

4. Next the material parameters for the bodies have to be entered. Select for example **body 1** → select **Set material values** *Fig 1.13* → A dialogue opens where the parameters ϵ_r and σ can be entered. Double clicking on the input field *Body name* enables to change the default name “body 1” to an arbitrary name.

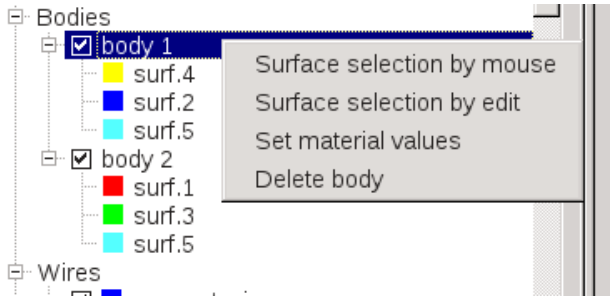


Fig 1.13: Defining the bodies from the surfaces

5. The remaining step to be done is to specify the type of the each surface (Fig 1.12):

   Select **Surfaces → Metallic and dielectric surfaces (bodies)** → A window as shown in Fig 1.14 opens where to enter the boundary condition to be

applied to all surfaces.

- *Metallic boundary*

  Always the PEC part of a body boundary. Caries two current distributions: one contributes to the the internal fields of a body and the other one contributes to the external field of the body.

- *Dielectric boundary*

  Both tangential components of the EM fields are matched here; again with have two current distributions.

- *Free space*

  If there is a surface in spree space, i.e. outside the specified bodies it would be given this property, meaning metal in free space (EFIE problem with a single layer current distribution). Such a surface is not allowed to bound a body.

- *In body 1, In body 2,...*

  The user could think of a ideal conducting metal disk for example completely immersed into the material of body 1 or body 2, respectively. Again, such a surface is not allowed to bound a body.

In Fig 1.14 the blue and red top surfaces are assumed to be ideal conducting metal plates of type *Metallic boundary* and all other ones are of type *Dielectric boundary.*

Surfaces can be layered by geometrically thin sheets, see Fig 1.15. Selecting *Add layer* enables to enter the material properties.

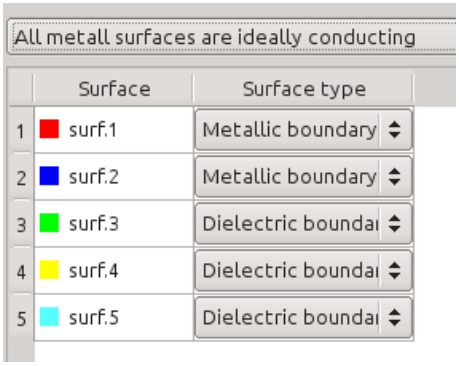


Fig 1.14: In the example case the top plates are of metal whereas all other surfaces are dielectric interfaces.

Note	Layers are only possible on surfaces of bodies. Layered surfaces must be of type *Dielectric boundary.* For details see Secs. 3.1, 3.1.6,4

Based on the given information the user is invited to carefully study the examples under $CONCEPT/examples/parallel-resistors and $CONCEPT/examples/Three-cubes.

In case of a body that is positioned completely inside another body with no common boundary (the user could think of two sphere with the same center but different radii) further information needs to be entered:

(right click) **Body in body** → **Assign** → a window opens where to enter which body is inside another body. In the example $CONCEPT/examples/Bodies_in_bodies a structure is treated where we have a spherical body inside a cylindrical body which again is completely inside a cube. In addition we have an open PEC cone inside the cube, not in touch with the other bodies.

The user must be very careful specifying the “body in body” information: CONCEPT-II does not check if the data is correct.

By default all bodies are immersed in free space, which is body 0. The topic is further explained under Section 3.1.4.

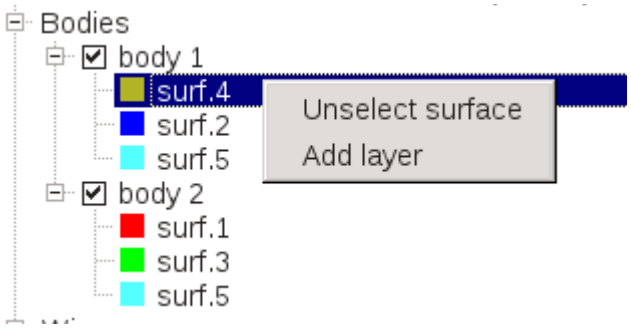


Fig 1.15: Adding a layer to surf.4. Clicking on *Add layer* open a dialogue where to enter the material parameters and the thickness

1.9.2.3 Wires

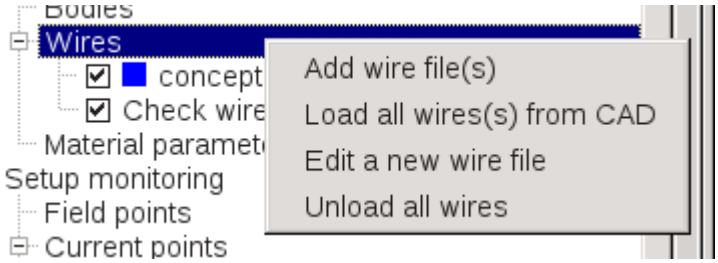


Fig 1.16: The drop-down list related to wires

Various features exist for wire treatment, see Fig 1.16. Similar to surfaces, wires may be organized or grouped in different files which should begin with “wire.” or end with the suffix “.wire”; *wire.1* or *concept.wire* are examples for valid names.

Add wire files(s) → existing ASCII files in the *CONCEPT-II* specific data format (see Sec.3.10) can be loaded into the simulation.

Selection of **Edit a new wire file** opens a dialogue window where all required data such as the file name, number of wires, coordinates, radii and number of basis functions (number of wire segments) can be entered.

Note	Deactivation of the check marks in the small boxes in front of the loaded wire files or surface files let the corresponding structure parts disappear in the display area. This does not mean that they are deleted from the simulation!

It is important that the thin-wire assumptions are not hurt, notice the corresponding remarks under Sec. 1.6.1. Basic errors in wire modeling are detected once the check box *Check wires* is active. Wires with errors are marked red, warnings appear in pink.

For each wire the number of segments (basis functions) has to be entered.

- It is recommended that 3 segments are specified on electrically short sections which appear under the following circumstances:
  - Wire-grid modeling where surfaces are simulated by corresponding wire grids; each wire being electrically short in general.
  - Short wires which connect different surfaces. Such wires are important from a network point of view to model an existing galvanic connection.
  - Short pieces of vertical wires, connecting lines to ground.

- Rules for discretization have already been given Sec. 1.6.1.

- If it turns out that the initial discretization of a wire is not sufficient when increasing the frequency in a frequency loop the number of basis functions

will be automatically increased according to the specified number of basis functions per wavelength.

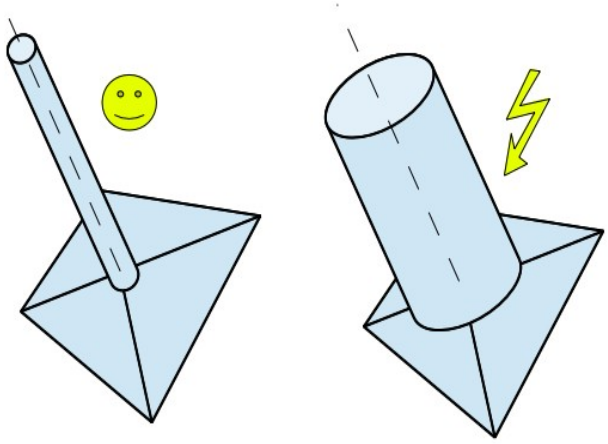


Fig 1.17: Wires attached to a surface patch node common to four triangles.

Left: the ratio of radius over edge length of the triangles is ok.

Right: the radius of the wire is far too large compared to the edge length of the surface patches. This is already a problem which would require a 3D surface patch grid. Note that in case of a wire a simple line current is flowing along the axis, testing the boundary condition is not accomplished around the circumference.

Wires can be connected to the surfaces patch nodes of metallic surfaces and to other wires. Connections are recognized if the difference in coordinates is smaller than 10^{-5} of the applied unit (default is meter) . A triangular patch is allowed to have one or more wires attached to a single edge, hence there must be always 2 edges between different points of attachment in a triangular grid. Wire identification numbers are written into *concept.out* . By these numbers can be assessed if a galvanic connection has been recognized or not.

As explained by means of Fig 1.18 it is not allowed that wires intersect with each other somewhere along their extension. In case of a physical point of attachment straight wire section must start or end at that point.

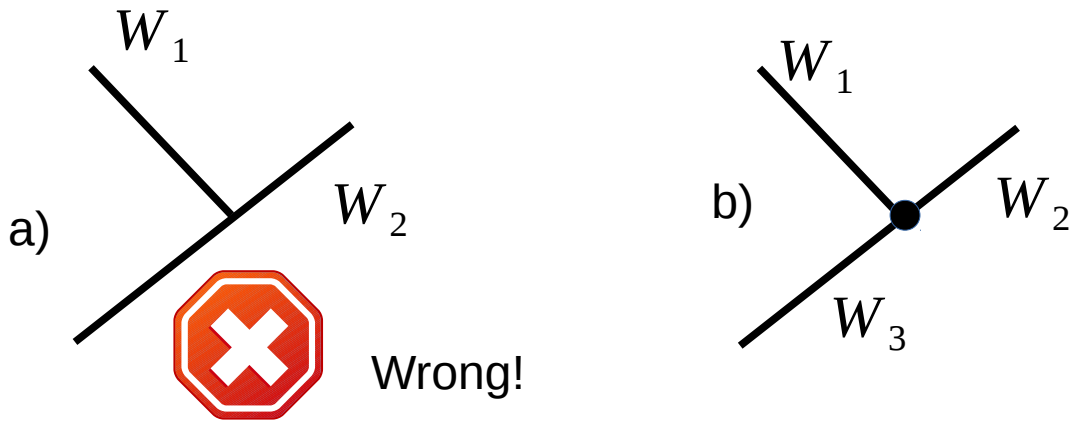


Fig 1.18: In a wire junction wires must start or end

a) Wire 1 intersects with wire 2, a junction is not recognized. An error message is given

b) If wire 1 shall be calvanically attached, three wires are necessary meeting in the common point of attachment

Both surfaces and wires can be loaded by lumped elements R, L, C in various combinations.

Select a wire file in the simulation project tree → (right click) → **Set loads** → dialogue window opens for the specification of loads (possible positions: beginning, middle and end of a wire), various predefined circuit elements are available. An example is demonstrated in Fig. 1.19: we have a series connection of a resistor and an inductor ,R,L positioned in the center of wire 1, 2, ….

In case of a large number of loads:

- **Append** the corresponding number of lines (“Total loads”), click **OK**
- Right clicking into the 'Wire no.' field provides the possibility to set all wire numbers at once.
- The same holds for all other positions. Right clicking on 'Circuit identifier' gives **Set global.** Activating gives a selection of various circuit identifier. The chosen identifier can be assigned to all wires.
- Loads can be taken from files. Format: frequency real part imaginary part (three real numbers in each line)
- For inserting, deleting or copying of a line ('Load 1', 'Load 2',... , see Fig 1.19) right click on 'Load 1' or 'Load 2' or whatever load number shall be selected

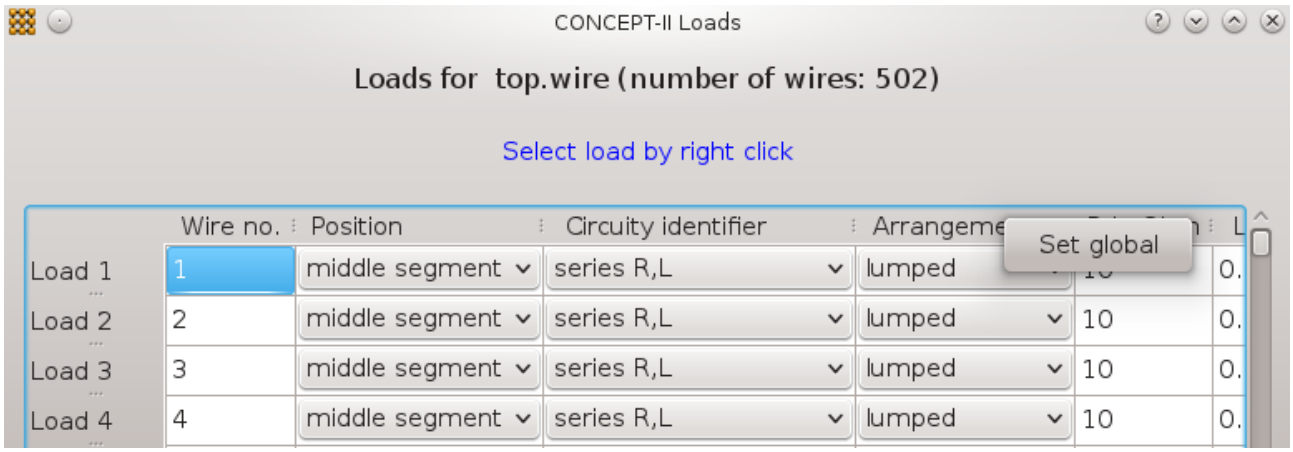


Fig 1.19: Dialogue window for specifying lumped loads

For placing lumped loads on surfaces (at least one surface files must be loaded):

right-click the project tree entry → **Load(s) on patch edges** → **Set edge loads** → *a* dialogue window *opens.*

Select an edge between two patches by a right click. A load symbol is located at the chosen edge, specify the circuitry identifier and the R, L, C values.

For voltage probes: select **Voltage probes on patch edges** by a right click → **Set Probes**→ select the probe at the required position.

1.9.2.4 Frequency sampling

In Fig 1.20 all main entries of the **Setup simulation** section of the project tree are illustrated (**Frequencies, Excitation, Solver**) . Right-click on **Frequencies** gives the shown drop-down list.

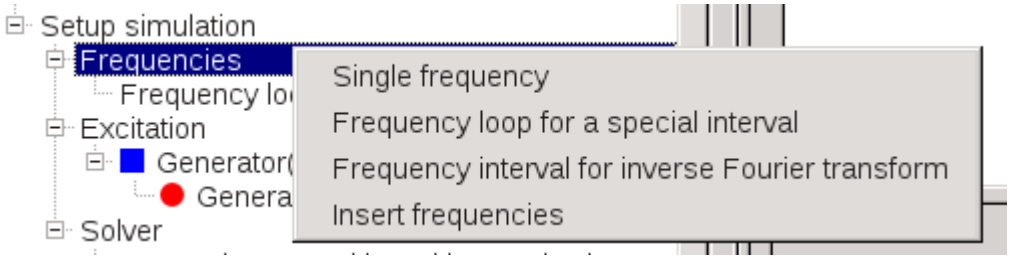


Fig 1.20: The **Setup simulation** section of the simulation project tree

Note that a special frequency sampling is required if an inverse Fourier transform shall be applied for time-domain system responses.

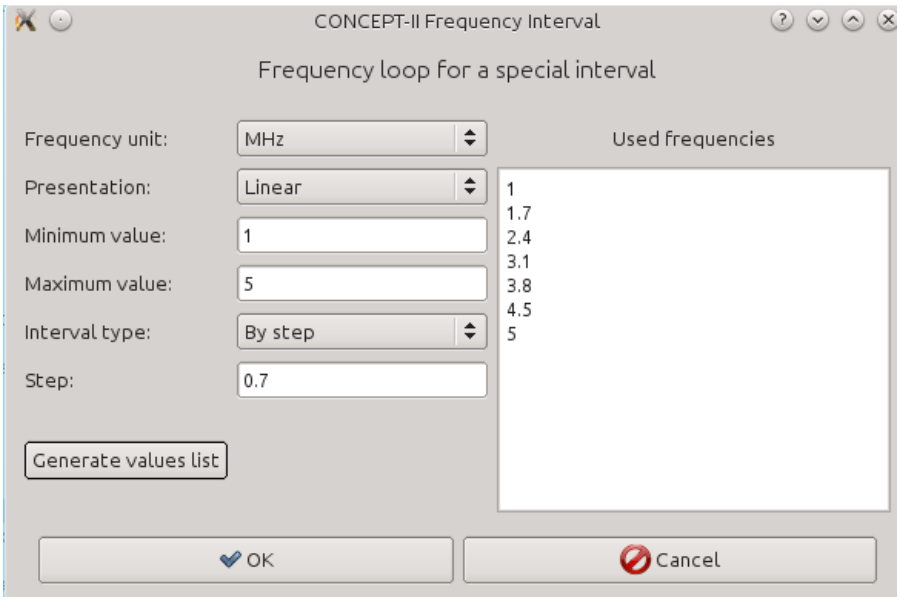


Fig 1.21: Window for setting the frequency sampling

Clicking on **Frequency loop for a special interval** → right click **Set values** opens a window for specific entries, see Fig 1.21. Clicking on **Generate value list** gives the list in the 'Used frequencies' area. It is possible to add values “by hand” here or to edit an existing list of frequencies. In the example the interval between 1 MHz and 5 MHz is sampled by 0.7 MHz steps. Note that the last step comprises only 0.5 MHz. Click on **OK** to activate the selection.

The next logical step when setting up a simulation is to specify the type of excitation. According to Fig 1.22 a number of responsibilities are existing.

CONCEPT-II offers the possibility to compute time-domain system responses based on an inverse Fourier transform IFT. A careful selection of the frequency interval and of the frequency sampling is required; details can be found in Sec. 3.8 .

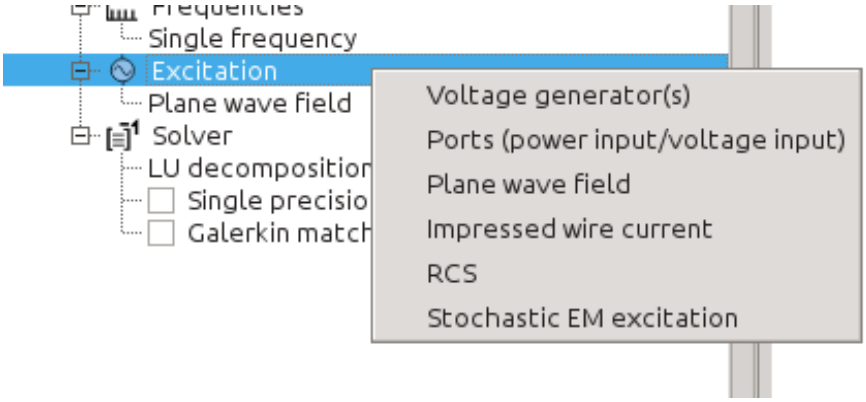


Fig 1.22: Excitation variants

How to use **Stochastic EM excitation** is not yet described in this manual. The

results of an application have already been published:

M. Magdowski, A. Schröder, H.-D. Brüns, R Vick: Effiziente Simulation der Einkopplung statistischer Felder in Leitungsstrukturen mit der MoM, EMV Düsseldorf, 2014, Seiten 238-244.

Simulation of coupled-in quantities in a mode stirred chamber.

1.9.2.5 Voltage generators

Voltage generators can be located both on wires and surfaces (edge generators).

To specify one or more voltage generators the following steps are necessary:

Generators on wires

Excitation → **Voltage generator(s)** → Generator(s) on *wire_file_name* (example *antenna.wire)* → ***Set generator(s)*** → a window according Fig 1.23 opens.

Position the mouse pointer on the wire where a generator shall be located. Right click selects the wire and a new line appears in the input window. Next set the position of the generator. Three locations are possible: **1st segment**, **middle segment**, **last segment**.

Of course we can also enter a voltage generator by appending a line (click the **OK** button, Fig 1.23) and entering the wire number by hand. The wire number is shown directly under the display area when placing the mouse pointer accordingly.

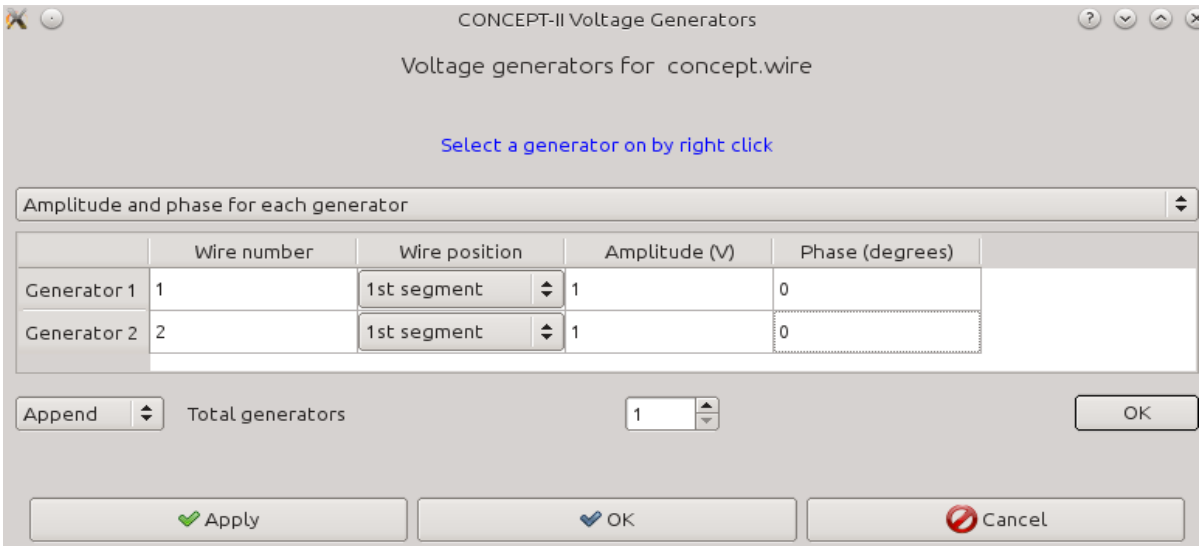


Fig 1.23: Window for the specification of voltage generators on wires

Delete a generator: Click on entry 'Generator no.' at the beginning of the corresponding line →the line is marked. Right click on Generator no. → **Delete**

Voltage generator on an edge between adjacent patches (edge generator)

Select item **Generator(s) on patch edges** in the project tree → right click → **Set edge generator(s)** → click on the required edge of the surface in the display area. For a change of polarity right-click on the same edge again. Specify voltage magnitude and phase in the window that has been opened.

1.9.2.6 Ports

In practice it is frequently of interest to compute only the mutual coupling of a system at feed points (voltage or power generators), located at different positions of the structure under investigation – one could imagine a ship carrying a number of monopole antennas. Such feed points are designated as ports. As demonstrated in Sect. 3.2 the mutual interaction between ports can be described by a reduced admittance matrix of order *K*, where *K* is the number of ports, which have been defined. In general *K* is much smaller than the order N of the system matrix.

Right mouse click on the item *Excitation* in the project tree → *Ports (power input/ voltage input)*

If both wires and surfaces are in the simulation the following project tree entries appear (wire_file_no: name of the wire file):

- Port(s) on wire_file_1
- Port(s) on wire_file_2

…

----- Ports on patch edges

Right-Clicking on *Port(s) on wire_file_1* → *Set values* opens a dialogue window as depicted in Fig 1.24.

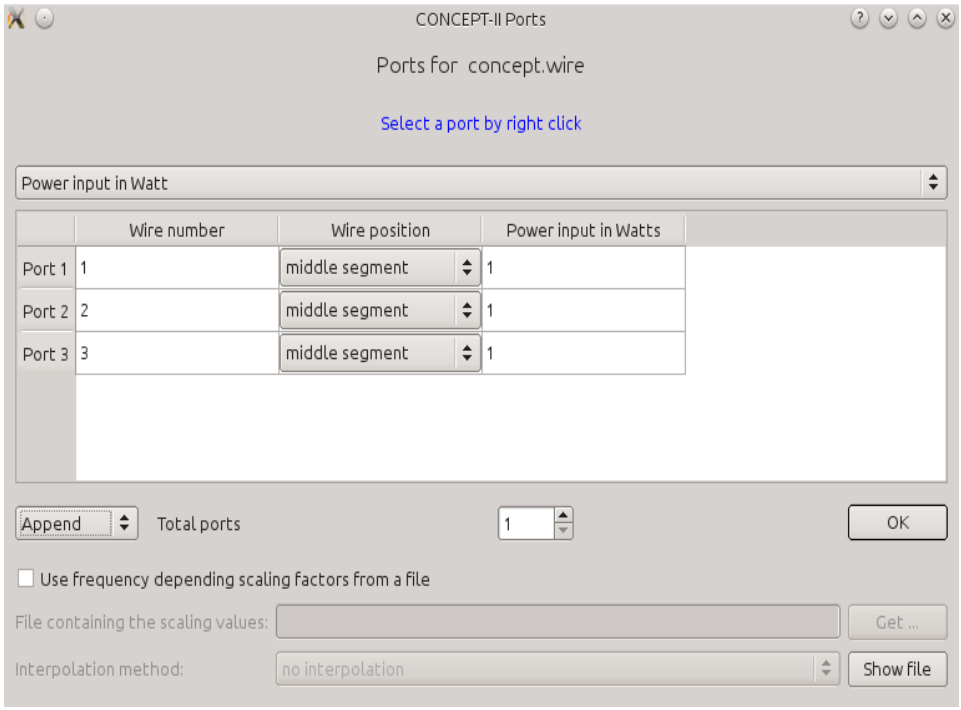


Fig 1.24: Example of three ports, each defines a power generator in the center of a wires 1,2,3 of the the wire file concept.wire

Note	Note that generators (power or voltage generators) do not excite the structure 'at the same time' in case of ports. The tools under **Post processing** provide the possibility to select which port is the active one. All other ports are short circuited. Loads which may be defined at the position of ports are not effected.

Note: appropriate wires can selected by right-clicking on them in the display area. Then choose the required location (middle, last, 1st segment) then.

In case of an edge generators choose *Port(s) on patch edges → Set edge port(s)* → a dialogue window opens. Select the required edge for a voltage or power generator by a right mouse click. Change generator polarity by a second click on the same edge.

Defining ports enables the computation of S-parameters between the ports. In addition to this a file named *red-y-mat.asc* (various formats available) is generated containing the complex admittance matrix for each frequency step. Each matrix is describing the coupling between the ports. In the example case according to Fig 1.24 we have three ports, hence the admittance matrix will be of order 3x3.

For further details refer to Sect. 3.2 .

- The sequence of the generators including the corresponding edge coordinates and reference directions are written into the file *patchgen.in* (default name).
- It is important that adjacent generator symbols of a common feed region are pointing into the same direction (example: generators around a cylindrical section)!

1.9.2.7 Plane wave field

Setting all angles to zero provides a plane wave propagation against the z axis with the E vector directed in the positive direction of the x axis. The E vector can be rotated around the k vector by the polarization angle Ψ.

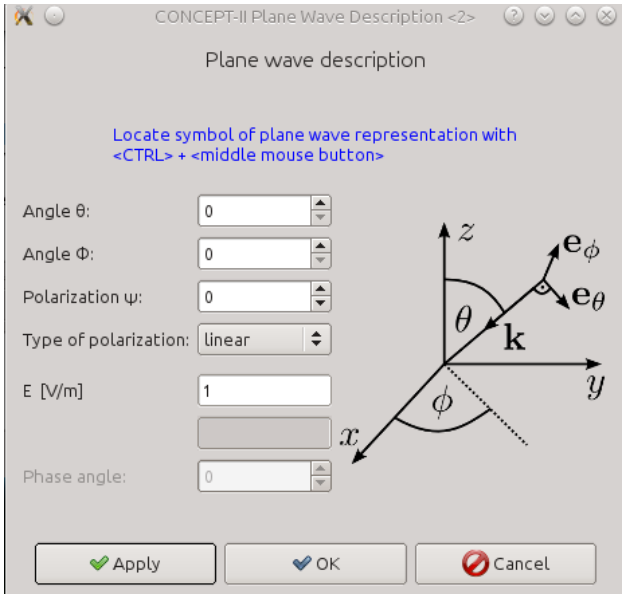


Fig 1.25: Input fields for defining a plane wave excitation.

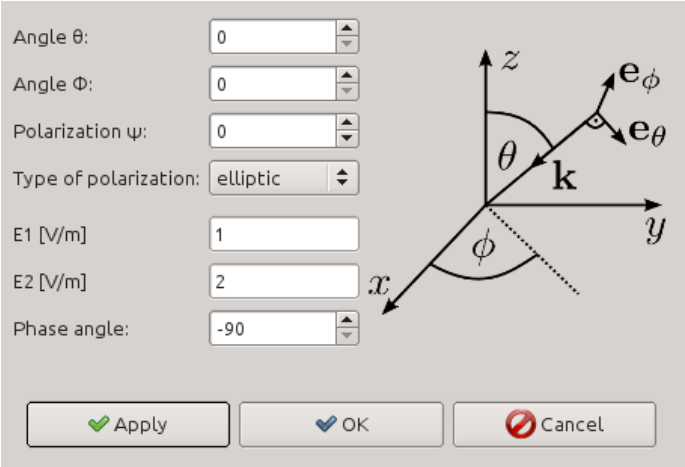


Fig 1.26: Data input for elliptic polarization

For elliptic polarization we have the following initial position:

The vector with with amplitude E1 is pointing in positive x direction and the vector with amplitude E2 is pointing in the negative y direction. Both vectors are perpendicular to each other. The phase angle, which refers to E2 determines the direction of rotation for elliptical or circular polarization.

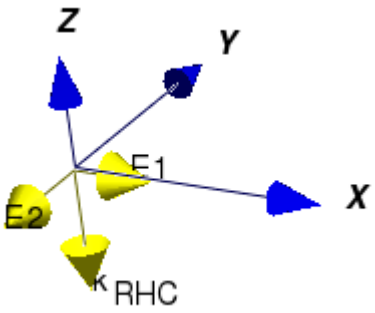


Fig 1.27: Orientation of the E field vector and the wave vector based on the values as given in

Rule: The sense of rotation is determined by rotating the phase-leading component towards the phase lagging component. The field rotation is observed as the wave is viewed as it travels away from the observer (looking into the direction of the wave vector k). The values according to Fig 1.26 lead to a situation according to Fig 1.13, that is we have a right circularly polarized wave (RHC: right hand circular polarization, clockwise rotation). Rotating the k vector into the positive z direction and the E1 vector into the positive x direction we have:

$E_x = E_1 \sin(\omega t - \beta z)$, $E_y = E_2 \sin(\omega t - \beta z + \delta)$ with δ as the *Phase angle*.

Assuming a positive phase angle the sense of rotation is left hand (LHC) .

1.9.2.8 Impressed wire current

Wires carrying line currents with known amplitudes can serve as excitation. The induced currents on the other surfaces and wires that might be present have no impact back onto the exciting wire currents. Hence these known currents are impressed ones.

A short section of a wire with known current may be used as a current source.

Long wires can be used to simulate the stroke of a lightning, either as a nearby stroke or as a direct stroke, based on the transmission line model originally proposed in [Uman], see end of Sec. 3.8.4.The channel is taken into account by a sequence of wires carrying an impressed current distribution.

Clicking on **Impressed wire current** (Fig 1.22) opens a dialogue window as depicted in Fig 1.28.

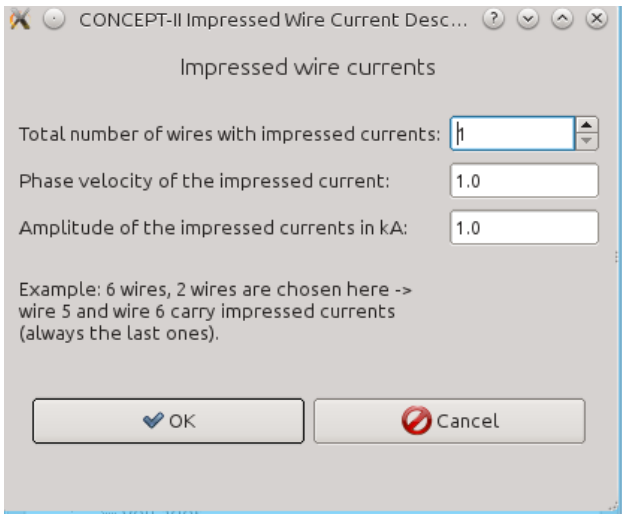

CONCEPT-II Impressed Wire Current Desc...
Impressed wire currents
Total number of wires with impressed currents:
1
Phase velocity of the impressed current:
1.0
Amplitude of the impressed currents in kA:
1.0
Example: 6 wires, 2 wires are chosen here ->
wire 5 and wire 6 carry impressed currents
(always the last ones).
OK
Cancel

Fig 1.28: Dialogue for specifying impressed wire current

- *Total number of wires with impressed current*

  Entering a number n means that the last n wires carry an impressed current distribution. In the example we see a “1” in the top spin box. Hence the last wire has a known current distribution to be specified in the remaining two input fields.

  If only a single wire has been entered the corresponding EM fields can be computed, see **Solver** in the simulation project tree, item **Compute only the impressed field.**

  If a short piece of wire with an impressed current is part of a loop it is possible to model an ideal current generator.

  Wires with impressed currents should form a sequence (end of a wire is connected to the beginning of the next one).

- *Phase velocity of the impressed current*

In case of a lightning simulation the relative propagating speed of the time-domain current function.

0 → All basis functions have a constant amplitude all over the wire(s) with an impressed current distribution. Useful to model a current generator.

1 → Propagation with speed of light (0.333 is frequently taken for lightning simulations).

- *Amplitude of impressed current in kA*

  For more information refer to Sec. 3.8.4

1.9.2.9 Solvers

The last section of the simulation project tree concerns the selection of the solver, Fig 1.29. The number of processors to be used can be entered in the upper left

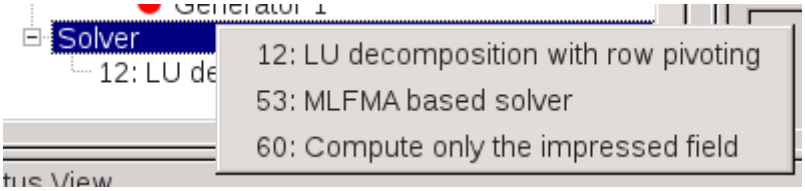


Fig 1.29: Solvers in the CONCEP-II package

corner of the main window, right to the green light button.

Details with respect to memory and computation time have already been mentioned in Sec. 1.5 and Sec. 1.7.

For *LU decomposition with row pivoting* it is advantageous to select the number of CPUs according to a quadratic number: 4, 9, 16, 25,....This rule does not apply to MLFMA.. Never enter a number larger then the number of CPUs available on the computer!

Note	Application of the ACA or the MLFMA solver: the typical dimension of the structure under consideration has to be electrically large. Triangular meshes are preferred!

ACA is currently only available for single CPU usage and pure surface patch structures in free space. LU decomposition is the default solver which can be applied to any kind of structure.

A major limitation of the MoM for electrically large objects is the very large

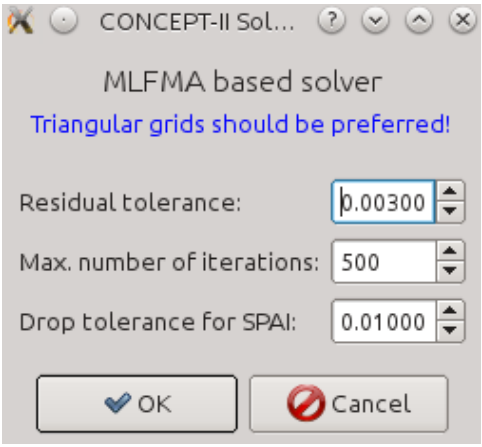


Fig 1.30: Default entries for the MLFMA solver

computation time and the huge amount of RAM space that is necessary. The MLFMA reduces both storage and CPU time requirements significantly to N log (N). In case of the ACA we have an $N^{4/3}\log N$ proportionality. The MLFMA solver can only be applied to metallic structures including wires and surfaces and symmetry.

Residual tolerance: Determines the precision of the iterative numerical solution. 0.01 could be sufficient in case of far field computations. 0.0001 could be necessary for almost closed cavities and complex structures.

Max. number of iterations: If the *Residual tolerance* is not achieved after this number of iterations the computation stops.

Drop tolerance for SPAI: Determines the sparsity of the preconditioner. The smaller the number the more dense the preconditioner and the faster the convergence. Disadvantage of a small number is a long set-up time.

In the case of plane wave with no structure being present or in the case of an impressed current distribution with no further structure parts, the item **Compute only the impressed field** *can* be selected.

1.9.2.10 Precision of the numerical solution

The default matching technique is line matching which is fast and gives good results for freely radiating antennas/structures. Galerkin matching provides even more precise results with the disadvantage of requiring a higher matrix set-up time.

Cases where a high degree of precision is required:

- Very low frequencies, typical structure size 1m and below
  - No problem in case of pure wire structures, see $CONCEPT/demo/example1-wire-loop for example.
  - Problem to compute the proper (very small) real part of the input impedance in case of loops where both wires and patches are involved. No problem at higher frequencies. No problem in case of pure patch loops or monopole antennas modeled by strips of patches (triangles).
- Cavities with small apertures, where coupled-in or coupled-out field quantities are of interest: use Galerkin matching; three integration points are normally sufficient (Fig 1.31)
  - ➔ No problem with wire antennas inside or outside the cavity and the wires being not attached to patch nodes.
  - ➔ Problem in case of wire antennas that are attached to patch nodes. Workaround here: model monopole antennas by small patch strips and use edge generators. The following equation gives the width w of the stripes: w= 4 r with r as the radius of the original wire.
- Single precision: Experience has shown that the single precision mode only gives valid results for well-conditioned problems with external excitations in conjunction with iterative solvers (e.g. MLFMA). For direct solvers or problems involving cavities use double-precision instead (default).

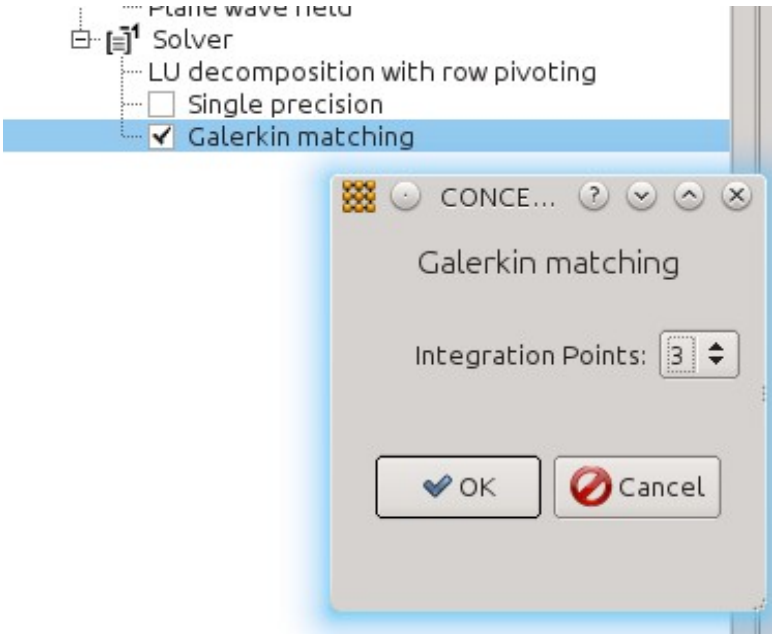


Fig 1.31: Galerkin matching can only be used in case of a pure triangular mesh

1.9.3 Post processing tools

When clicking on the **Post processing** tab, the 'Project View' section looks as depicted in Fig 1.32. All programs for post-processing purposes can be found here. For the application of these programs the *CONCEPT-II* back end must have been finished.

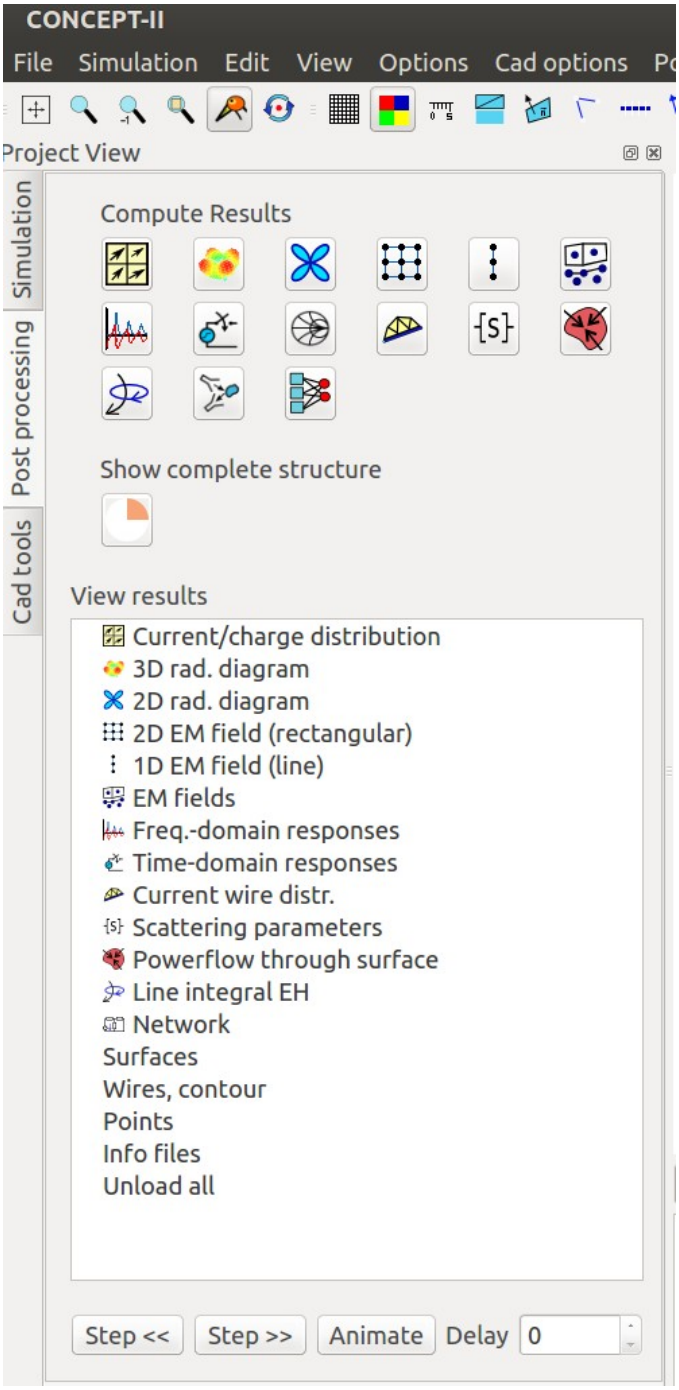


Fig 1.32: The post processing project view and project tree, respectively

Under 'Project View' 15 buttons arranged. Clicking on a button opens the

corresponding dialog window. Tool tips give information about the purpose of each tool.

→ Representation of current/charge distribution both on wires and surfaces.

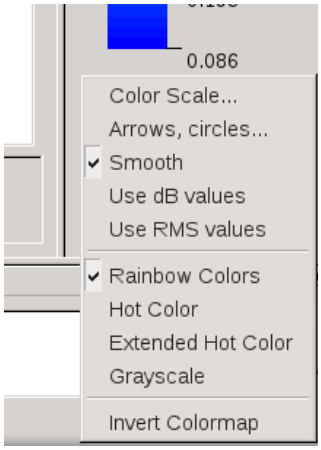
At least 16 phase intervals between 0,...,360 degrees are recommended for a movie of the current distribution. If the movie is running to fast enter an integer value (10,20,... depending on the graphics card) into the *Delay* spin box at the bottom of the **Post processing** section.

Features:

Scale widget: surf.0 is selected → the color map in the scale widget right to the display area refers to the surface current distribution

Default representation is current vectors combined with colors on the patches. The colors are indicating the magnitude.

Scaling: Right-click on the *Scale widget* (section right to the *Display area*, which automatically opens) → A number of items can be selected for modification of the displayed data. Changing the scaling has only an impact on the data belonging to the actual surface, indicated in the drop-down list in the head section of the *Scale widget.*



Color scale → Default color map is **Rainbow Colors**. The red and blue marked surface areas can be extended (the color red is shown between a value to be chosen and the maximum value. Blue colors indicate small values.

Different color maps such as 'Rainbow colors' (default), 'Hot colors' are available.

Arrows, circles... → Scaling of vectors, surface current distribution, field distributions.

Smooth → Show only a single color per patch in turn-off mode. When hovering the mouse pointer over a patch the value of the depicted physical quantity is shown underneath the *Display area*.

Note	The contents of the vertical value list right to the color bar as depicted in the *Scale widget* may change a little, depending on the mode of representation.

dB values → Current distribution in dB scale

Clicking on **Arrows, circles...** opens a dialog according to Fig 1.33 .

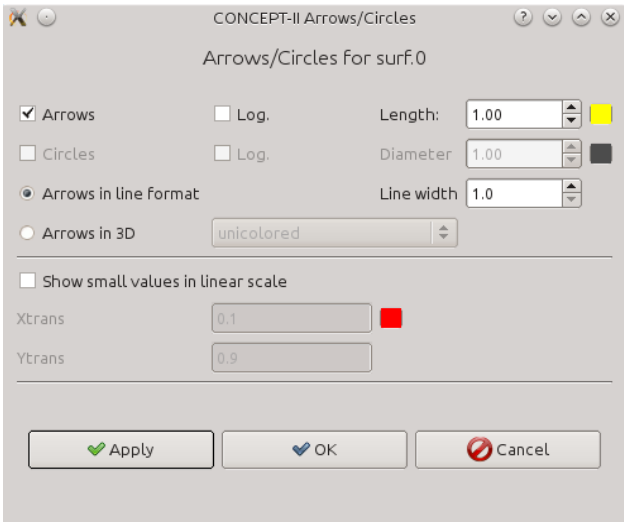


Fig 1.33: Dialog for the modification of vector representation

Tip	The mentioned features for vector representation of surface currents apply also to the vectors of electromagnetic fields.

Scale widget: wire.0 is selected → the color map in the scale widget right to the display area refers now to the wire current distribution.

Scaling: Right-click on the *Scale widget* → As has been described for surface current distributions a number of items can be selected for modification of the displayed current distribution.

A black arrowhead indicates the current direction at the beginning of the wire. In case of a change of sign in the current distribution an arrow with a different color appears at the corresponding location pointing in the reverse direction.

Scaling: Right click into the scale widget → **Arrows circles** → Lengths, widths, and colors of the arrows can be changed

For displaying the current distribution on wires as curves click on in the **tool bar.** The following window pops up:

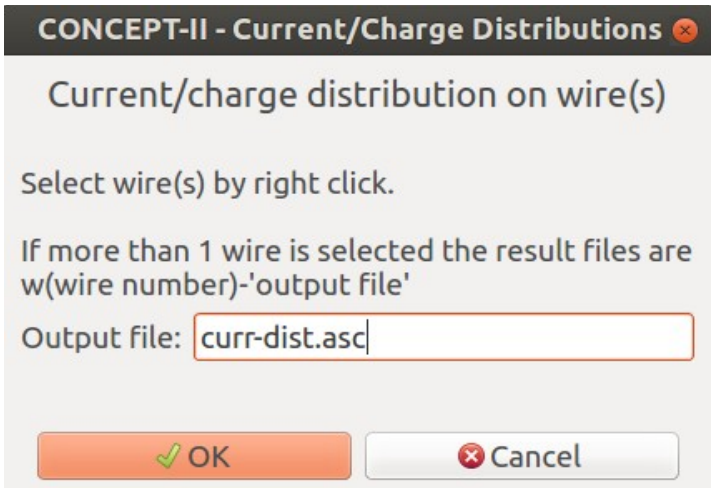

CONCEPT-II - Current/Charge Distributions
Current/charge distribution on wire(s)
Select wire(s) by right click.
If more than 1 wire is selected the result files are w(wire number)-'output file'
Output file: curr-dist.asc
OK
Cancel

Fig 1.34: Wire current distributions as curves

Right click on the wires whose current distribution are to be shown. ASCII data are available, see header of the respective files, for example *w1-curr-dist.asc*, *w2-curr-dist.asc*... Clicking on **OK** provides the curve(s) in a gnuplot preview window. For more display possibilities right click on the *Current wire distr.* item in the *View results* tree. The *CONCEPT-II gnuplot front end* opens. Among other features .png and .svg graphics files can be generated.

→ Dialog window for 3D radiation diagrams opens

- Electric far field, EM near field distribution, magnitude, theta components, phi components
- The field values are computed on a spherical grid in a $\Delta\theta, \Delta\phi$ region with a number of steps to be specified in each direction
- Complex far field data for E are stored
- Movies of 3D radiation diagrams developing as a function frequency

→ Display 2D radiation diagrams

- Electric far field in orbit plane of arbitrary direction. The orientation of the plane can be chosen arbitrarily – the corresponding angles are explained in the window.
- The diagram plane can be displayed together with the radiating structure
- Magnitude, theta and phi components, LHC and RHC, gain in the case of port excitation (power input).

→ Computation of 2D electromagnetic field distributions in rectangular flat areas.

Subsequent dialog opens for field representation. Movies are possible.

- E field, H field, Energy flux density, specific absorption rate

→ Computation of electromagnetic fields at an arbitrary position in space.

Observation points can also be set along straight paths in space or can be taken from a file.

In case of a frequency loop fields are computed for all frequency steps.

→ Computation of EM fields.

If the check box 'Fast parallel field computation' is active a fast computation of field plots of electrically large areas is possible under the following conditions (status Oct. 14):

- No wires and only triangular patches are present
- Free space, structure not over ideal ground
- No symmetry planes specified
- At least 2 CPUs are necessary

'Treatment' ACA → the adaptive cross approximation is used to speed up the solution. Details about the technique have been published here:

A. Schröder, H.-D. Brüns, C Schuster: Fast Evaluation of Electromagnetic Fields using a Parallelized Adaptive Cross Approximation. IEEE Transactions on Antennas and Propagation, vol. 60, no. 12, pp 6058-6061, Dec. 2012

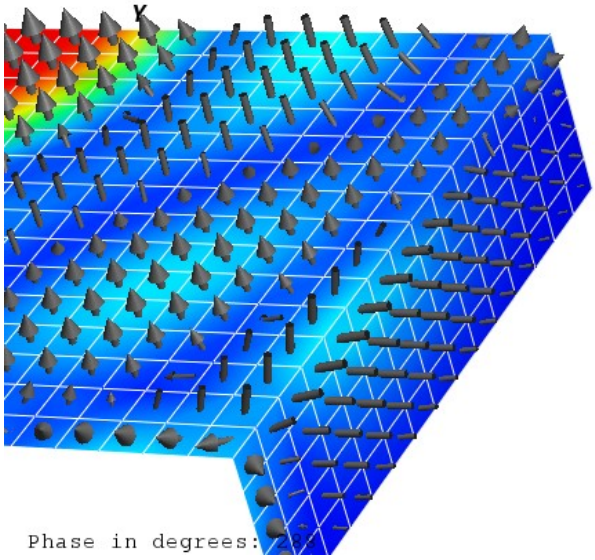


Fig 1.35: E field distribution with 3D vectors. Note that the corresponding phase is indicated in the lover left corner of the *Display area*

Drop down list with 3 entries:

- **Field probes on a surface**

  Any *CONCEPT-II* surface patch file may serve as input file. The probe positions correspond to the centroids of the patches. A subsequent dialog opens, asking for data that are necessary for displaying the result. Specifying an appropriate 'Number of phase intervals' it is possible to generate a movie. Further action: set the mouse pointer on the 'Scale widget' and right click, then click an **Arrows, circles...** and do the vector scaling. An example of the computed normal electric field distribution on a metallic surface is illustrated in Fig 1.35

- **Fields along a path**

  A *CONCEPT-II* file in wire format may serve as input (see Section 3.10)

Note	The beginning and end coordinates of each wire are taken as field points! Hence at least 10 wires per wavelength are necessary in the high frequency case.

- **Field points from a file**

  File format:

  first line: x1 y1 z1 , second line: x2 y2 z2 , ….

Clicking on **Compute** starts the field computation. The field points are shown as small spheres, the color indicating the field strength value. Setting the mouse pointer on a sphere make the field strength value appear under the display area.

Provided that that the check box 'Compute field values for all frequencies' has been activated one may step through all frequencies.

Clicking on the **Plot of EM fields** icon (see *Tool bar*) opens a dialog where to specify a file name. Set the mouse pointer on a field point and right click; the selected point becomes white. The data belonging to the selected point are written into the specified file. The gnuplot front end is used to show the result.

The described proceeding is also possible for the entries **Fields on a surface** and **Fields along a path**

Clicking on (see *tool bar*) enables the user to compute EM fields along a path.

→ Plot system responses as a function of frequency. A drop-down list shows the system responses that are available.

Examples: **E(f) (Probes from monitoring)** → Field point have already been set in the simulation phase before the back end has been started

E(f) (Post processing) → Fields have been computed after the back end run, tool:

→ Enables the analysis of a transient system excitation, i.e., computing time-domain system responses. Precondition is that a special frequency loop has been chosen: **Simulation → Frequency → Frequency interval for inverse Fourier** transform.

Note	In case of an IFT: Both the limits and the proper sampling of the covered frequency interval are important for a successful application. Computation may be time-consuming as a number of frequency steps are necessary, depending various parameter such as typical dimansion of the structure, signal duration and rise-time.

→ Allows the representation of an antenna input impedance in the form of a Smith chart (program *smith*). Also representation of port impedances.

→ The current distribution on wires can be displayed. Currently only of importance for a sequence of time steps in case of an inverse Fourier transforms. Here the the corresponding wires must have been selected under **Simulation** (front end run).

→ Plot of S parameters in case of a port excitation. Definition of ports:

Simulation → Excitation → Ports (power input/ voltage input)

→ Computation of the power flow through a surface (numerical integration of the Poynting vector over a surface). The simplest way to create such a surface is to click **Cad tools** → icon **Sphere** . Only triangles are allowed for the integration surface!

→ Computation of line integrals of E or H, see Section 3.4.

→ The results of an RCS calculation can be displayed (monostatic radar cross section)

→ EM fields when port excitations are modified

To display the whole structure under investigation click on the **Build-geometry** icon . Depending on the kind of magnetic symmetry the icon appears differently. Full icon color means no symmetry has been specified.

Of importance for post processing purposes is the project tree which can be seen in the 'View results' area:

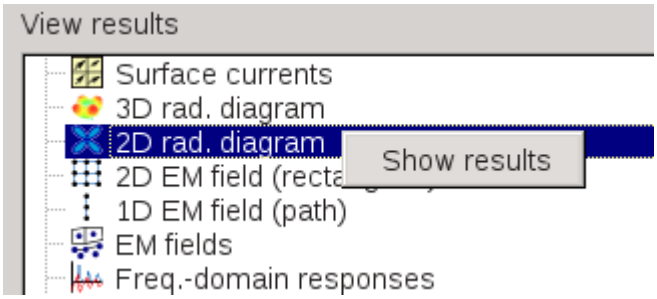


Fig 1.36: Handling of the post processing project tree

Right-click on an item, then click on **Show results.** This will load previously computed data into the display area, which will be a 2D radiation diagram in the example according to Fig 1.36. Right click on **Freq.-domain responses → Show results** displays the last generated curves again providing the possibility of tuning for an external representation for example (the *CONCEPT-II* Gnuplot front end opens).

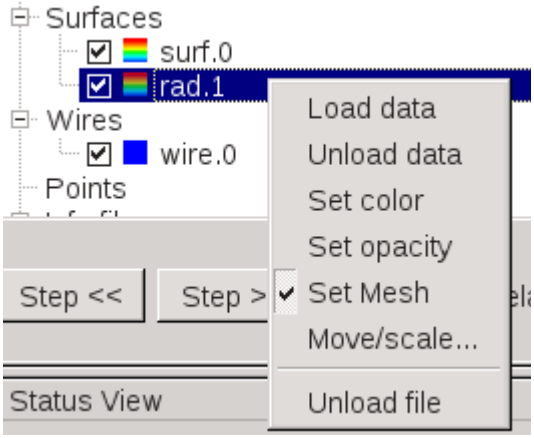


Fig 1.37: Drop-down list that opens on right mouse click

A 3D radiation diagram (if has already been computed) can be loaded again for example and scaled for a proper representation together with the structure including its surface currents. Once a 3D radiation diagram has been loaded, the entry **rad.1** (file with surface patch format) appears under **Surfaces** in the lower part of the P**ost processing** project tree (Fig 1.37). **surf.0** and **wire.0** are the default names of the loaded structure geometry. These files are automatically build from the input files as were specified under the **Simulation** section. Generally

surf.0 includes the combined surfaces patch files and *wire.0* contains the wires of all wire files.

Clicking on **Move/scale...** opens a dialogue by means of which a structure scaling can be accomplished. Important: this does not change the structure itself; it is only for graphical purposes. Note further that a 3D radiation diagram is scaled into a cube of unit edge length, hence its size needs to be altered and/or shifted in most practical cases, when depicted together with the structure under investigation.

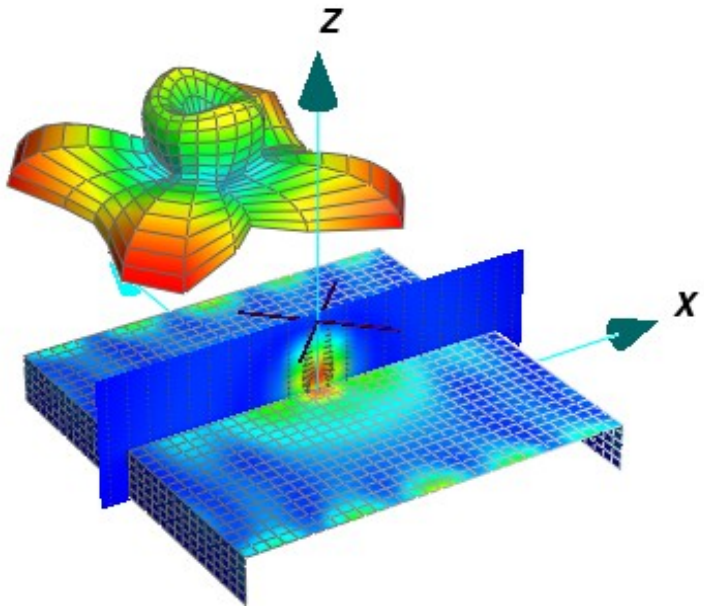


Fig 1.38: Antenna on a table with 3D radiation diagram and 2D field distribution

Fig 1.37 show an example where 3 different results are shown in a single picture.

Noticeable in the drop-down list according to Fig 1.37 is the item **Unload data** by means of which the data (surface current distribution for example) can be erased. Right click on the **Surfaces** item, see Fig 1.37 offers the possibility to load additional surfaces to be displayed in the display area. By means of **Load data** corresponding field or current distributions can be added (see Section 1.8).

2 Example: Top-loaded wire antenna on a table structure

The handling of the GUI shall be demonstrated by a simple example.

The input files of the following structure can be found under *$CONCEPT/examples/Table_antenna*. A top-loaded monopole antenna which is located in the center of a metallic platform shall be analyzed. The whole structure is placed on an ideal ground plane. The platform has a height of 0.5 m and consists of a top plate and two side plates being connected to the ground. The size of the top plate is 2m x 2m. The length of each wire is 0.75 m. There is an additional wire

between the top plate and ground modeling a coaxial feeding cable. The electrical and geometrical symmetries involved in the configuration shall be exploited and the frequency interval between 1MHz and 100 MHz is of interest.

The structure can be discretized with the *CONCEPT-II* CAD tools (notice that the **CAD tools** tab is active, we have *top.surf*, *side.surf*, *antenna.wire* and *coax.wire*). As the vertical antenna is fed, magnetic symmetry can be exploited (x-z plane and y-z plane). Hence there is only need to specify the geometry in the first quadrant of the Cartesian coordinate system as illustrated in Fig 2.1. In the example directory the ASCII file *Table_antenna.geo* is available for usage in conjunction with the *gmsh* tool, see Secs. 3.7 and 1.6. By this file the plates can be discretized outside the GUI. The corresponding data can be converted into the CONCPT-II surface patch data format (*Menu bar*, **Import/export**).

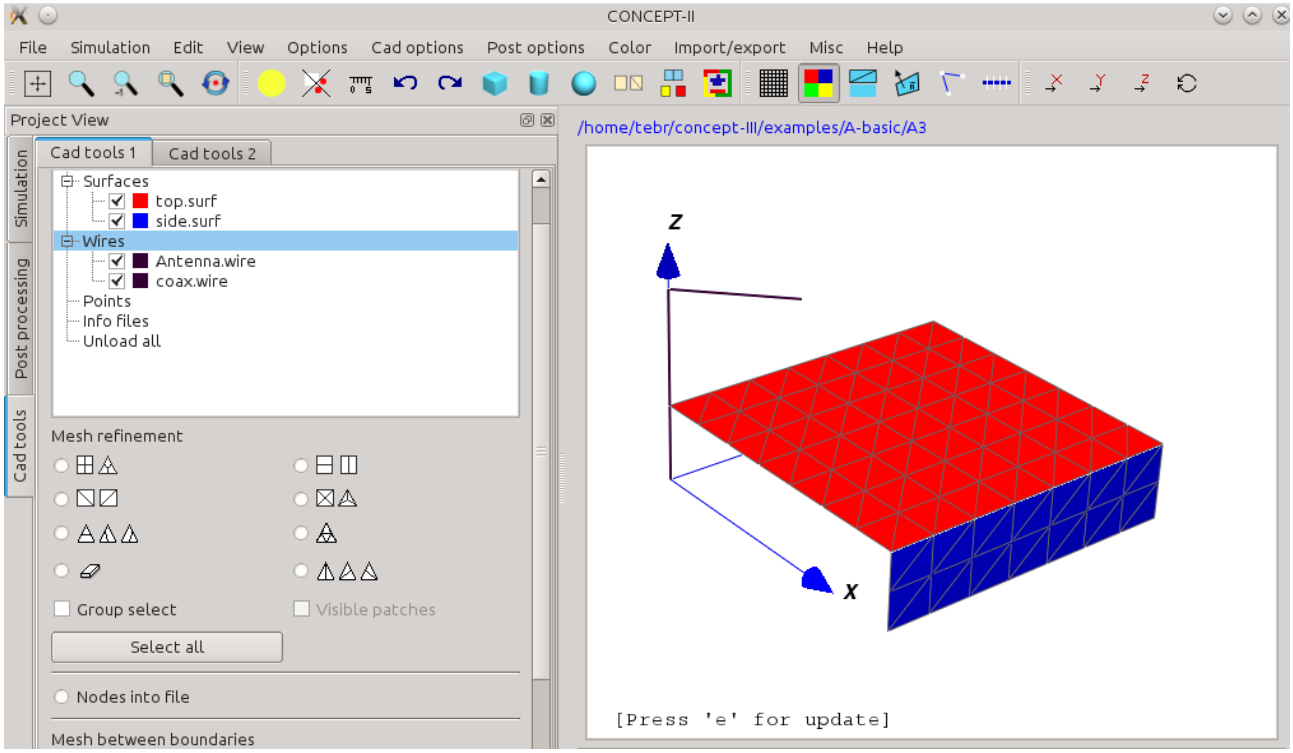


Fig 2.1: The top loaded antenna with supporting structure

Note	General rule: in the case of magnetic symmetry only the symmetric part of the structure needs to be discretized. The ideal ground plane always corresponds to the x-y plane.

Remember: a magnetic symmetry plane is characterized be E field vectors which

are completely tangential in the plane of symmetry. On the other hand the H field vector have only perpendicular vector components there. We observe a dual situation on an ideal ground plane which is assumed to be present in this simulation example (x-y plane).

2.1.1 The discretization of the surface part

We are working under the **CAD tools** section. There are different possibilities for discretizing the structure. One way shall be sketched shortly.

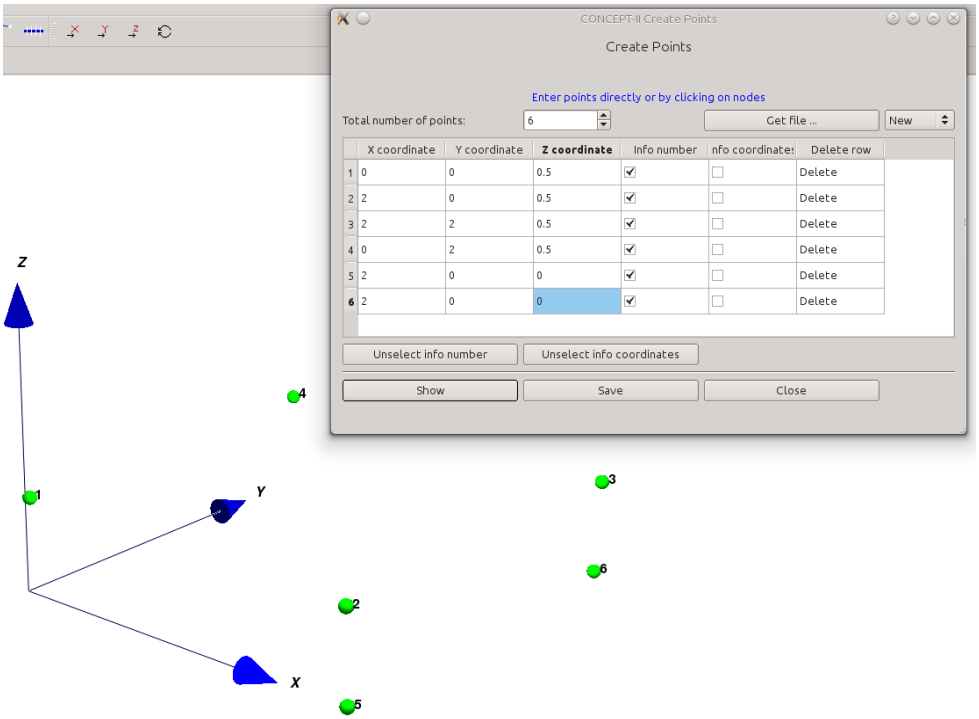
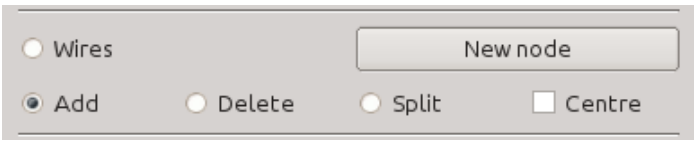


Fig 2.2: Corner nodes of the antenna table

- Activate the **Points** item in the cad tree → right click → **Edit a new point file** and enter all corner coordinates into the dialogue window which opens, see Fig 2.2.
- Activate the radio button *Plate, arbitrary quadrangles* and select the four points marking the top plate by right click. **Update** → a dialogue opens, enter the data as suggested in Fig 2.3. The name of the surface patch file has been entered (*top.surf*), also the frequency and the number of basis functions per wavelength. Note: for electrically small plates the number of segments might not be sufficient → correct manually under. *Meshes*.
- Next discretize the side plate and then enter the wires. For this purpose the following section under the **Cad tools 1** tab can be used:



The end coordinates of wire 1 correspond to the starting coordinates of wire 2. So the code recognizes a galvanic connection between them.

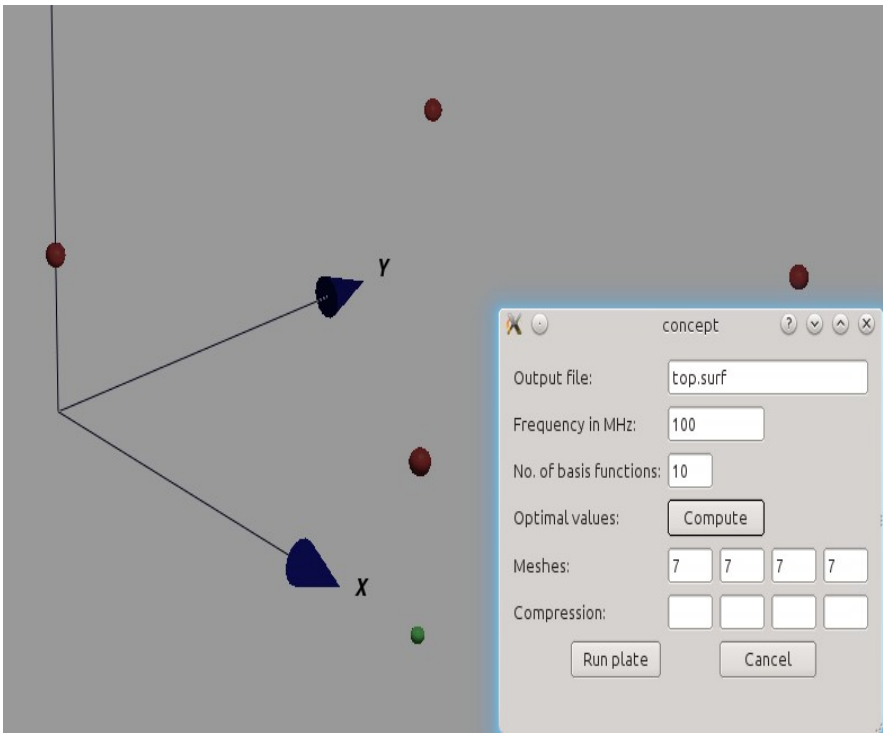


Fig 2.3: Meshing of the top plate. Notice the description of the various input fields

Next click on the **Simulation** tab (*Project view*) -> click on head entry **concept** → (right click) → **Load files fron 'CAD'**, continue to set up the simulation, moving through all menu items of the **Simulation** *Project tree*, starting at the top. According to Fig 2.5 all symmetry planes have been activated already (see check marks under **Symmetry**). This is also indicated by small yellow cylinders, appearing in the top left corner of the *Display area*. **Ideal ground** (always at z=0.!) has been activated, which is indicated by the following symbol: (change of

position: <Shift> + click on the mouse wheel). Select **Antenna.wire** and right click → **Edit wires** → a dialogue window according to Fig 2.4 opens.

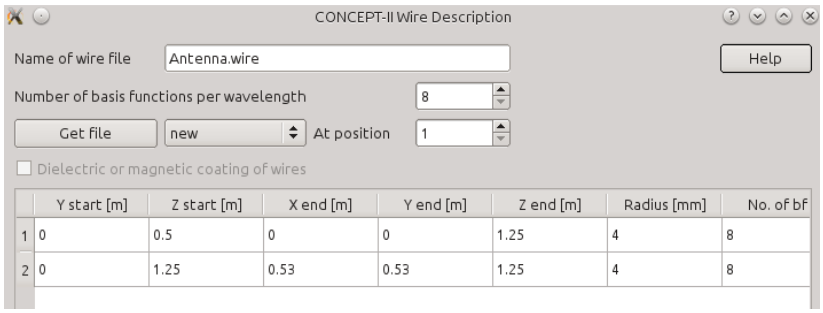


Fig 2.4: Dialogue window for editing wires

Right-clicking on a wire number appearing in front of each row provides the possibility of quickly inserting or deleting a line (a wire). For rapid global changes in case of many wires use the drop-down list left to *Total wires*.

A *Number of basis functions per wavelength* smaller than 8 is not possible. Eight basis functions on each wire are a good choice for the starting frequency of 1 MHz, λ=300m) where a wavelength dependent segmentation is crucial or impossible, respectively. For such low frequencies an electrostatic “point of view” is relevant. The user should answer the following question: “ How many charge blocks (approx. equal to the number of basis functions) are required for a good approximation of the normal electric field strength distribution. In other words: the surface charge distribution is approximated by kind of a stair-case function in the numerical simulation. From electromagnetic theory is known that we have a rapid increase at corners and edges. For a convergent current distribution we have the capture the integrated charge distribution in a sufficient manner.

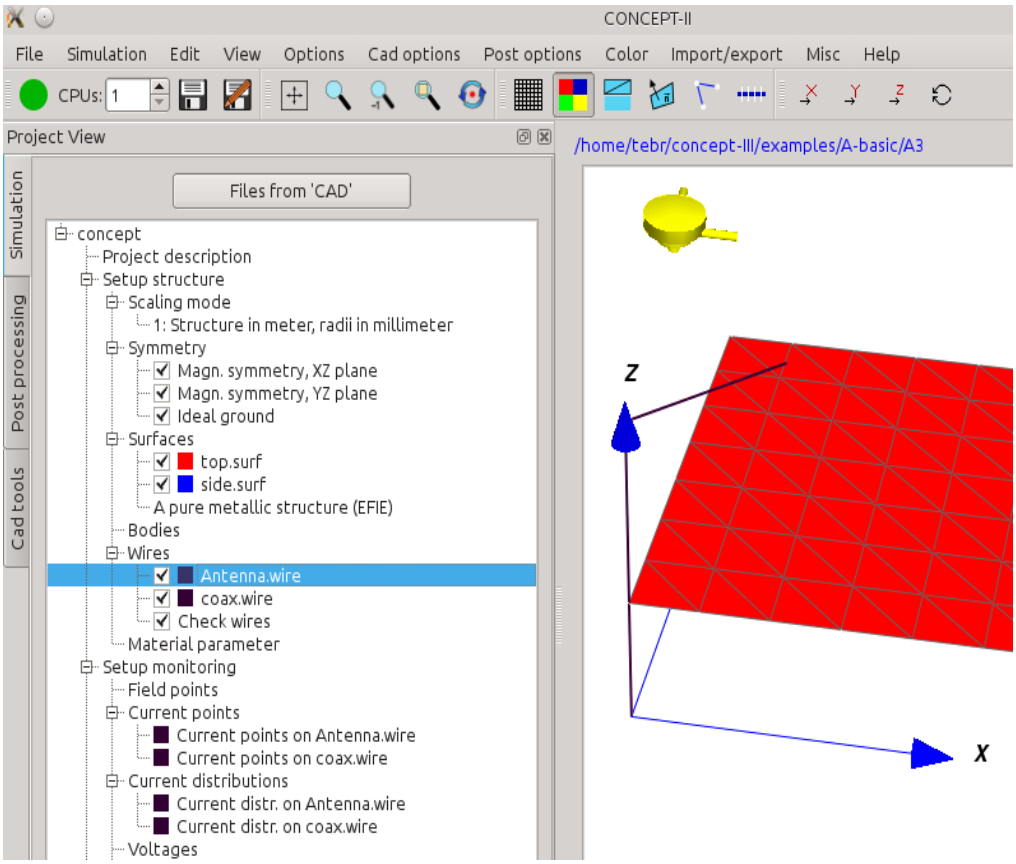


Fig 2.5: Status of the **Simulation** P*roject tree* after having loaded from the **CAD tools** section

The S**imulation** *Project tree* with all entries completed is shown in Fig 2.6.

Current in an observation point on a wire to be displayed as a function of frequency

Notice the blue dot at the vertical wire below the antenna table. **Current points** → **Current points on coax.wire** → right click → drop down list appears → **Set values** → dialogue window → **Append** → *Wire number* 1, *middle segment*

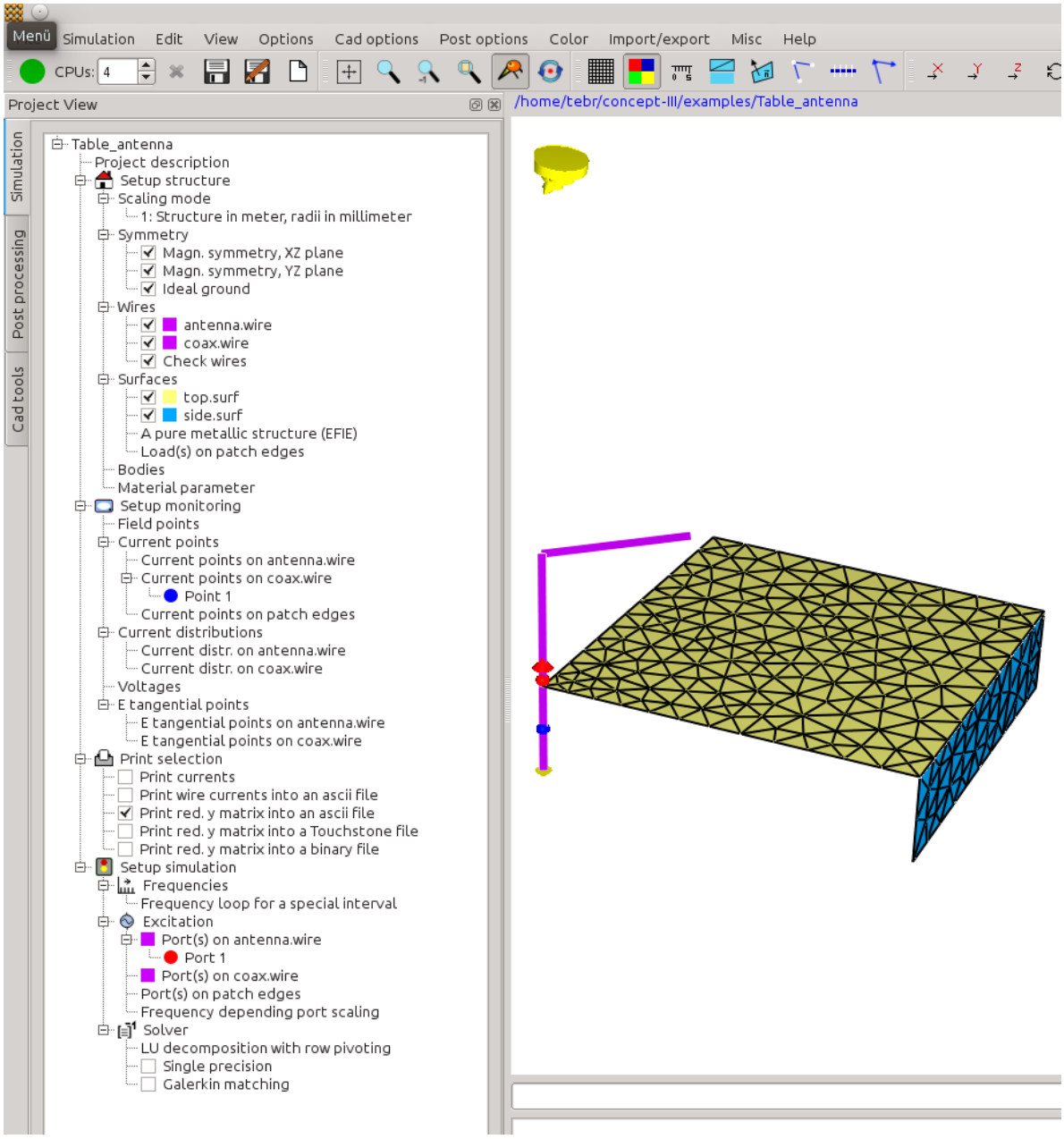


Fig 2.6: All entries of the simulation tree and structure as can be seen in the display area

Note

After a **Set values** window is open the corresponding wires can also be selected by right mouse click. The specific location (beginning, middle, end) has to be entered in the window itself.

Frequency sampling

A frequency loop shall be specified, 10,20,...,200 MHz: **Frequencies** → right click → **Frequency loop for a special interval** → a dialogue window opens →*Frequency unit* **MHz**, *Minimum value* 10, *Maximum value* 200, *Interval type* **By step** →*Step* 10→ **Generate list** → All frequency samples appear under *Used frequencies*.

It is important to note that only the frequencies that are explicitly listed are taken in the simulation after having clicked on **OK**. Steps can be added manually.

Excitation

A power generator providing 1 W forward power shall be placed at the beginning of the vertical antenna wire.

Excitation → right click → drop-down list →**Ports (power input/voltage input)** → **Port(s) on antenna.wire** → right click → drop-down list → **Set port(s)** → **Append** → *Wire number* 1, *Wire position* **1st segment**, *Power input in Watt* 1

Before starting the simulation the structure should be carefully inspected. Enter the number of CPUs (see left side of the *Tool bar*) and click on the green circular **Run** icon and wait for the back end to finish...

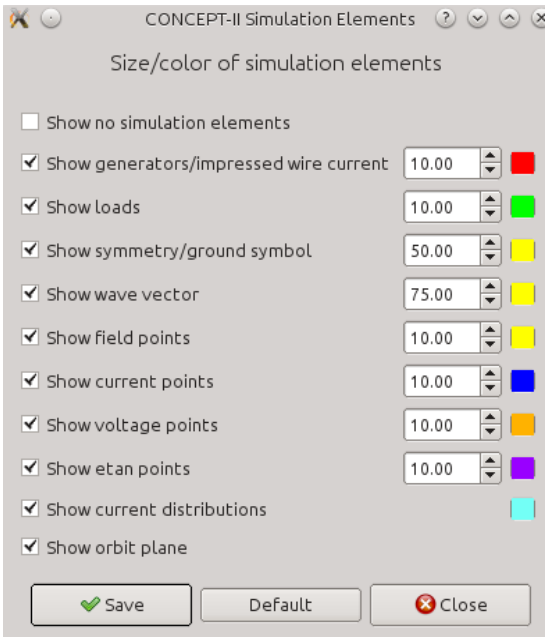


Fig 2.7: Window for the configuration of various elements that might appear in the display area*:*

Menu bar: **Options** → **Modify elements**

Clicking on the colored squares right to the spin boxes offers the possibility for changing the color of an element.

2.1.2 Post processing

The system current distribution is known after the back end has finished. It is a good idea to begin the post processing by displaying the structure current distribution for a first validation. Activate the **Post processing** tab.

2.1.2.1 Representation of the current distribution

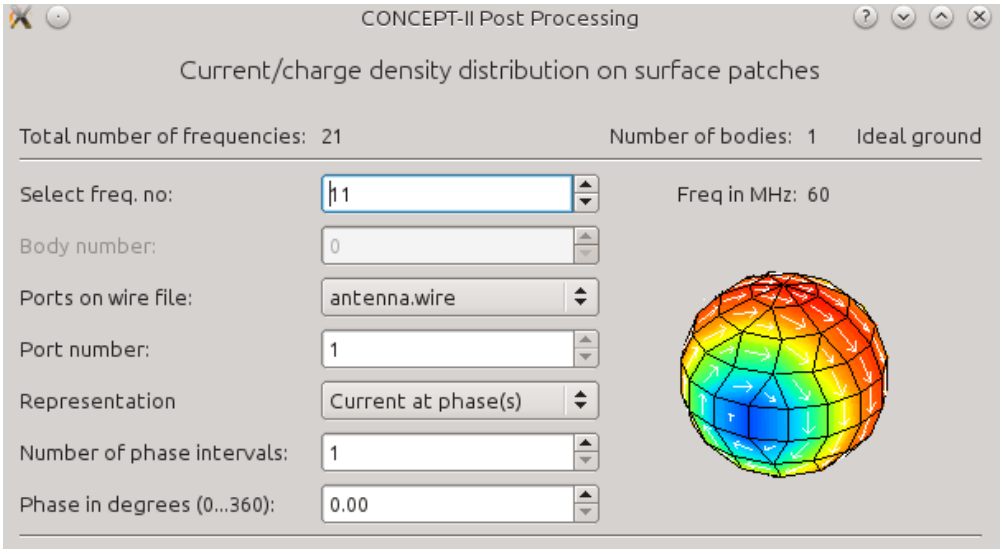

CONCEPT-II Post Processing
Current/charge density distribution on surface patches
Total number of frequencies: 21
Number of bodies: 1
Ideal ground
Select freq. no: 11
Freq in MHz: 60
Body number: 0
Ports on wire file: antenna.wire
Port number: 1
Representation Current at phase(s)
Number of phase intervals: 1
Phase in degrees (0...360): 0.00

Fig 2.8: Currents on surfaces and wires shall be displayed.

Click on , a dialogue window as depicted in Fig. 2.8 appears:

Number of phase intervals
For a movie 15, ...,20 phase intervals are a good choice

Phase in degrees (0...360)
Only active in case of a single interval.

An example current distribution for 60 MHz, 0° is shown in Fig 2.9. Moving the mouse pointer over patch of a plate provides the corresponding values of the current density in the output line under the *Display area* (. active, *Tool bar*). The colors are representing the magnitude of the current distribution for the chosen phase.

Some features:
Menu bar: **Options** → **Draw wires as...** → **Lines**
Options → **Wire line widths** → Enter value

File → **Save image as** → The contents of the *Display area* can be stored in a number of different graphics formats, an example is shown in Fig 2.9.

Right click on the Scale widget → a drop down list appear. Various color maps are available, default is **Rainbow Colors**. For vector scaling click on **Arrows, circles...** Fig 2.9 is drawn using the **Hot Colors** color map with *Arrows in 3D, unicolored.* Hot

colors are suited for black and white printing

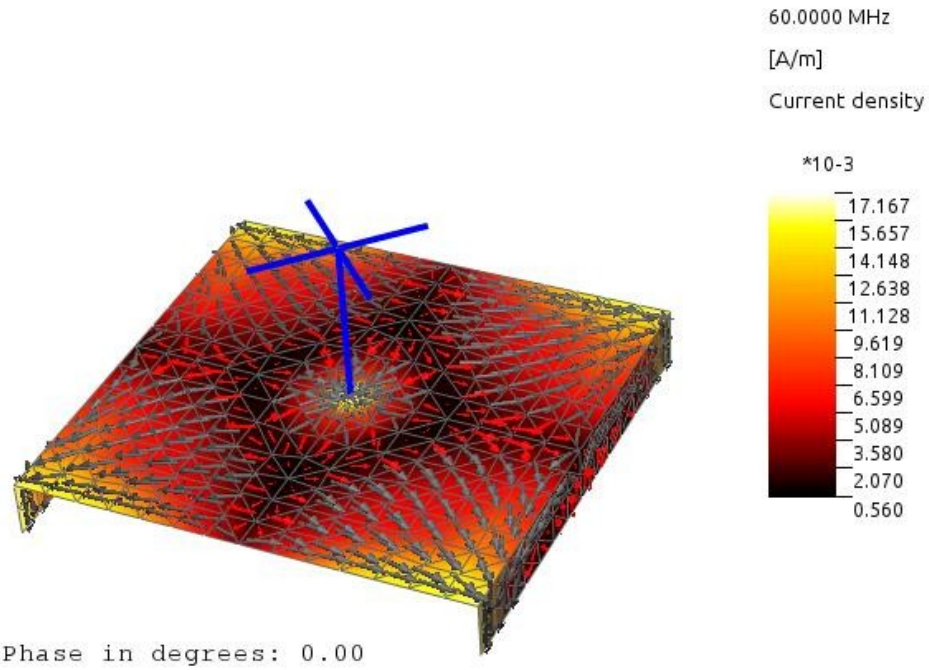


Fig 2.9: Current distribution on the example structure at a certain phase.

2.1.2.2 Representation of 2D electromagnetic field distributions

Click on this icon of the **Results Section**:

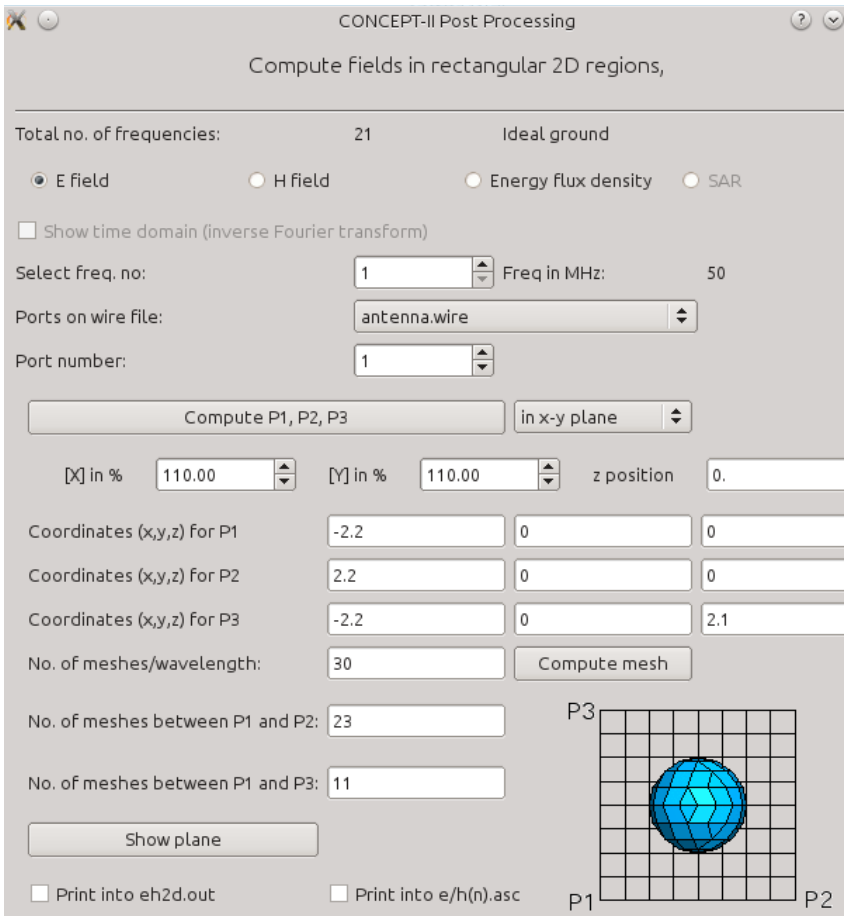


Fig 2.10: Compute electromagnetic fields in 2D regions

The following window appears (Fig. 2.10):
The rectangular plane is defined by points P1,P2,P3. Clicking on OK at the bottom of the window causes the E field strength to be calculated at all mesh nodes. In the dialogue window that follows the kind of representation, i.e. magnitude, resulting field strength vector etc. can be specified. Structure contour lines are taken from the files *wire.0, surf.0* in case of an intersection of the field observation window with the structure.

If there is need to repeat the representation choose the menu item **2D EM fields (rectangular)** in the post processing tree → right click → **Show results.** The stored complex field data of the rectangular area is used in this case, hence a time-consuming field computation is not necessary again.

The number of meshes that are necessary between the corner points of the 2D region depends on the frequency, the kind of representation and on the complexity of the field which is expected in the field observation window. Color maps require far more field points than vector representations. For the vector scaling right click into the *Scale widget* → **Arrows, circles...** For using *Show small values in linear*

scale notice Sec. 3.11. The **Post processing** project tree offers the possibility to show both fields and currents in the *Display area*. Structure parts can easily loaded or unloaded.

Warning

All field vectors may be perpendicularly orientated to the chosen field region. In such a case the field vectors may appear “randomly distributed”, the magnitudes are numerically zero and meaningless. Right click on the *Scale widget* → **Arrows, circles...** → activate *circles* or *Arrows in 3D.*

In the default representation circles are indicating the components that are normal to the field region.

Randomly distributed arrows may be observed at very low frequencies. Example: H field distribution of a short dipole antenna with a display region perpendicular to the wire axis, phase: 0 degree. The code might not be able to compute the very small radiation resistance properly. Note that *CONCEPT-II* is working in the frequency domain. Setting the phase to 1 or a few degrees gives a physical field distribution again in most cases.

2.1.2.3 EM fields along straight paths or at arbitrary locations in space

Click on

Here the spatial dependence of electromagnetic fields along straight lines in space can be computed after the *CONCEPT-II* back end has finished. Furthermore field points can directly be entered or the coordinates can be taken from ASCII files.

It is suggested not to set paths or field points too close to a surface – the minimum distance should be half a typical patch size. Otherwise there is danger that the piecewise constant surface charges become visible in the corresponding curve leading to kind of a step function. For precise data very close to an edge of the structure under investigation a local mesh refinement might be required!

When activating “Field point coordinates from file” the user will be asked for the corresponding file name. The data format is:

1st line x y z (coordinates of the first field point)
2nd line x y z (coordinates of the second field point)
3rd line etc.

In the case of a frequency loop the field computation is carried out for all frequency

steps at once .

Once the field computation has finished another dialogue opens where to enter the *Frequency number* and the total number of lines along which the distribution field components shall be displayed.

In case of a frequency loop click : the fields at all points can be plotted as a function of frequency. All points along the paths are numbered first followed by points that were explicitly specified by the user; an example is shown in Fig 2.11.

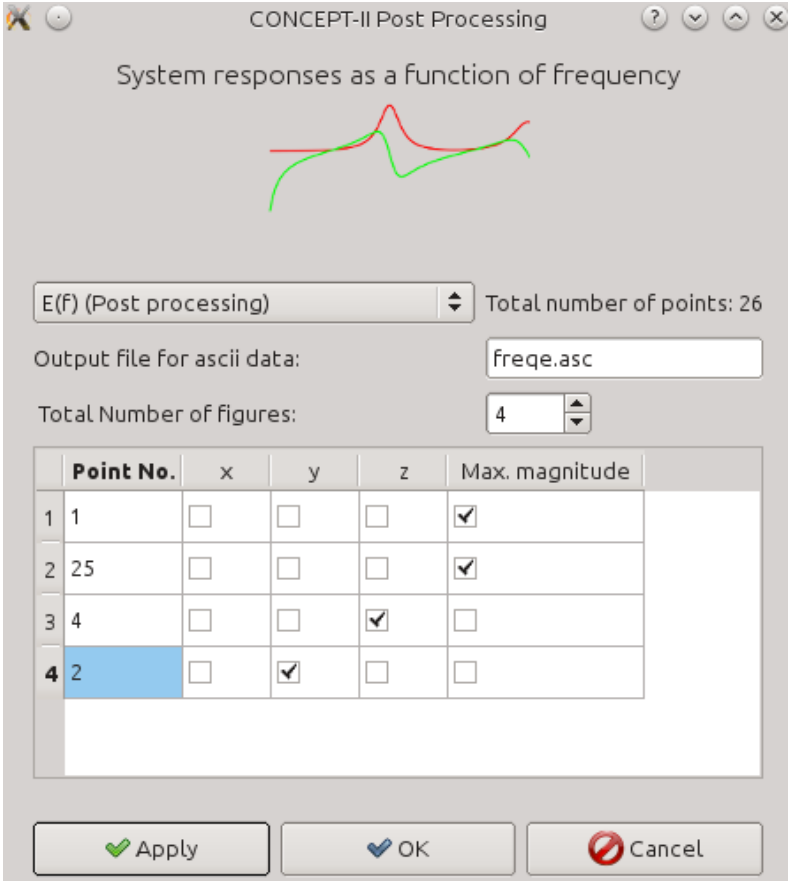


Fig 2.11: Dialogue window for specifying system responses as a function of frequency, here the components of the E field

Clicking on **Apply** plots the curves in the *Display area.*

In general the following quantities can be represented as a function of frequency: U, I, Z_{in} (Input impedance), E, and H. For port excitation only the input impedance of the first generator can be plotted here. The input impedance values are stored in the files *port-zin*n*.asc* as a function of frequency, with n as the port number. Use the gnuplot front end for representation (*Tool bar*).

Post processing tree: Select **1D EM field (lines)** → right click provides the items **Curve selection** and **Show results:**

> **Curve selection** → generate (further) curves along lines at a certain frequency to be displayed immediately in the *Display area*
> **Show results** → *the CONCEPT-II* gnuplot front end opens (see Fig 2.12) and the curves can be modified for subsequent external representation. Curves can be displayed in *CONCEPT-II* view (*Display area*) or gnuplot view. Formats available are .svg, .png, and .ps .

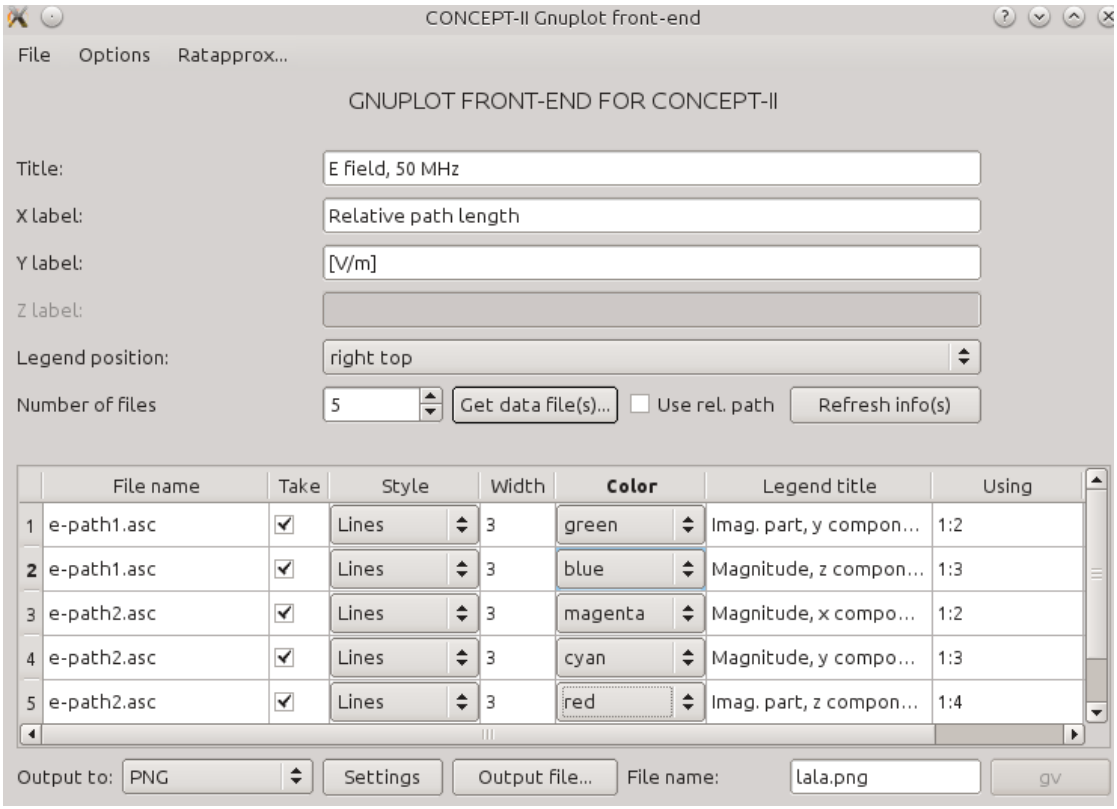


Fig 2.12: The Gnuplot front end of *CONCEPT-II*

Some features of the the Gnuplot front end

- Many curves in one plot are possible (for example: column 2 vs column 1 →enter 1:2).

- A curve is erased if the *Take* check box is deactivated.

- Data can be modified for representation: Example: $1:(\$2*1000)$ means that all data of the second column is multiplied by a factor of 1000. $1:(20*\log10(\$2))$ means that the values of the second column are displayed in dB.

- Hovering the mouse pointer over the *Using* input fields displays the head section of the corresponding ASCII data file, indicating the contents.

For further details see the Gnuplot documentation.

2.1.2.4 Creating 2D radiation diagrams

For 2D diagrams click on (**Post processing**)

A structure over an ideal ground plane gives a semi circle, in case of a free space problem the diagram plane refers to a full circle.

The necessary number of observation points for a good curve presentation depends on the complexity of the radiated field, which might include a lot of minor lobes. Field data (Maximum magnitude, gain, E_θ, E_φ ...) and complex E field data are stored in files, the names of which can be chosen arbitrarily. For the data format notice the head sections of the ASCII files.

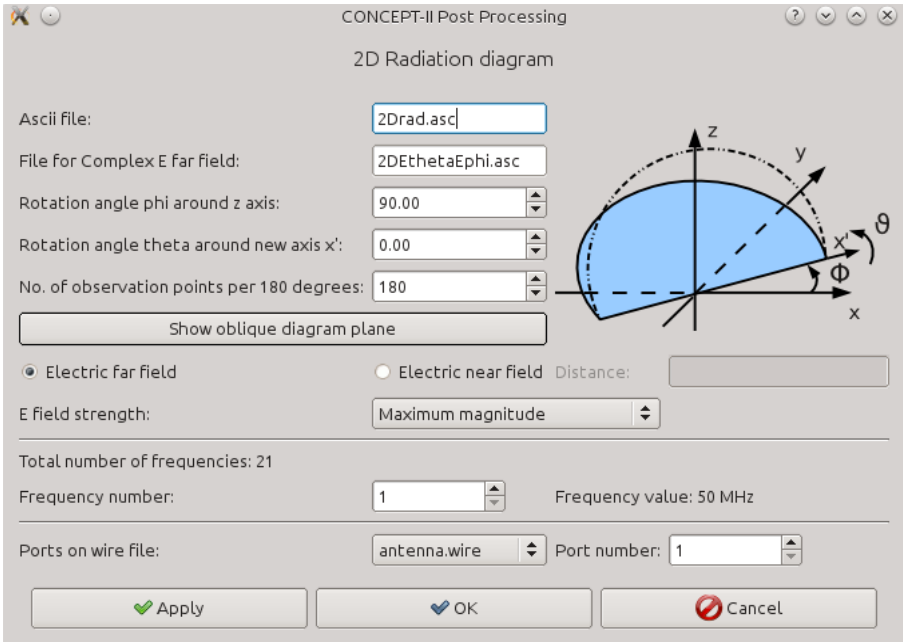


Fig 2.13: Dialogue window for 2D radiation diagrams

In case of a frequency loop data is stored into the files only for the selected frequency step, see input field *Frequency number*.
Clicking on **Show oblique diagram plane** depicts the diagram plane together with the structure under study. A rotation of Φ =90° around the z axis leads to a 2D radiation diagram in the y-z plane (for explanation please refer to the drawing in Fig

2.13)

In case of *Maximum magnitude* |E| radiation pattern data is given in the unit [V]. To get the E field strength value of a certain direction simply divide by the distance of the far-field observation point. For a near field computation the radius of the diagram plane must be entered, referring to the origin of the input coordinate system.

The following possibilities exist (see drop-down list *E field strength*):

- Maximum magnitude (of the polarization ellipse)
- Theta components
- Phi component
- LHC (left-hand circular polarization)
- RHC (right-hand circular polarization)
- Axial ratio
- Gain

LHC, RHC: A plane wave can be represented as a linear combination of a right circularly polarized wave and a left circularly polarized wave as known from theory. In case of a linear polarization the amplitudes of the LHC and RHC components are equal (curves lie on top of each other)
For linear polarization the maximum value for the axial ratio is set to 1000!

The gain is computed according to
$G = 4\pi \frac{U}{P_{\text{in}}}$ with P_{in} as the input power (excitation: ports, generators) and
$U = \frac{r^2}{2\eta} |\vec{E}(r, \Theta, \Phi)|^2$, η: impedance of free space. $\vec{E}(r, \Theta, \Phi)$ far-zone electric field intensity.

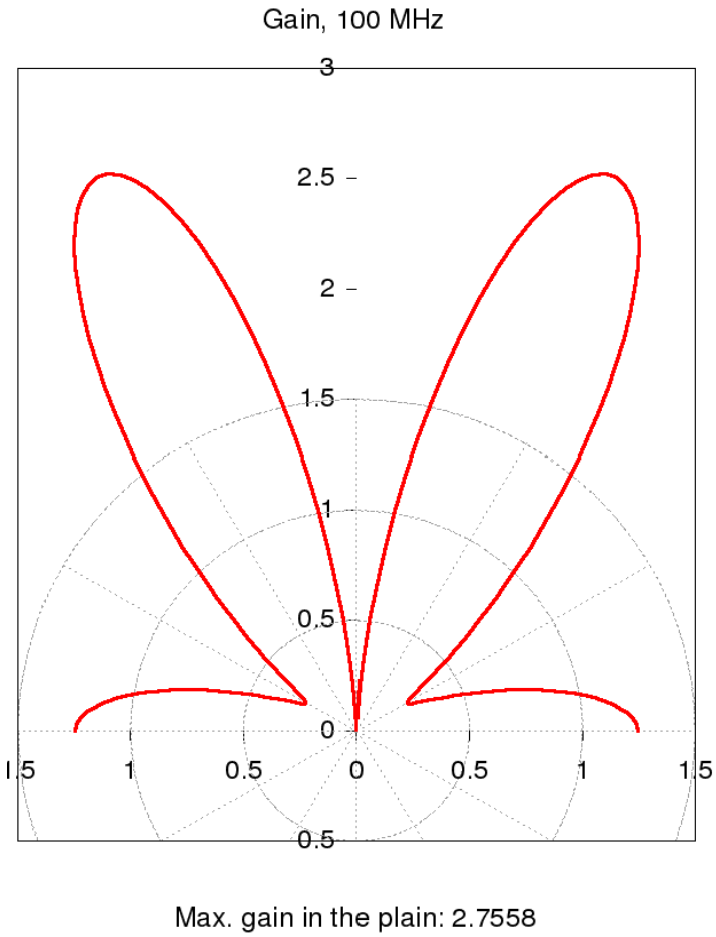


Fig 2.14: Gain of the table antenna at 100 MHz (y-z plane)

Fig 2.14 illustrates the gain of the top-loaded table antenna according to Fig 2.9 at 100 MHz

2.1.2.5 Creating 3D radiation diagrams

For 3D radiation diagrams it is necessary to specify a grid of field observation points on a fictitious sphere of unit radius. A appropriate spherical section can be defined by the angles “THETA” and “PHI”, referring to a spherical coordinate system (Fig 2.15).

Due to the electrical and geometrical symmetry of the antenna structure the azimuthal angle PHI could be restricted to the interval [0°, 90°]. Setting the “End angle PHI” to 90 degrees and selecting “Symmetry” → 'x-z and y-z plane' would automatically complete the 3D diagram. Under this condition the number of grid points in the phi direction could be reduced to one forth.

With increasing electrical size of the radiating structure a larger number of grid

points is generally necessary in order to have a diagram including all minor lobes which might appear. A general rule for an appropriate number of points cannot be given. Approximate rule: 20 X 20 elements for electrical small structures, 40 X 40 elements and more for electrical large structures. A 3D radiation diagram has surface patch format and always fits into a sphere of 1m radius.

Considering *All frequencies* the change in shape of the 3D diagram can be animated as a function of frequency.

Important data is written into *radiate.out* (ASCII file) the contents of which is shown

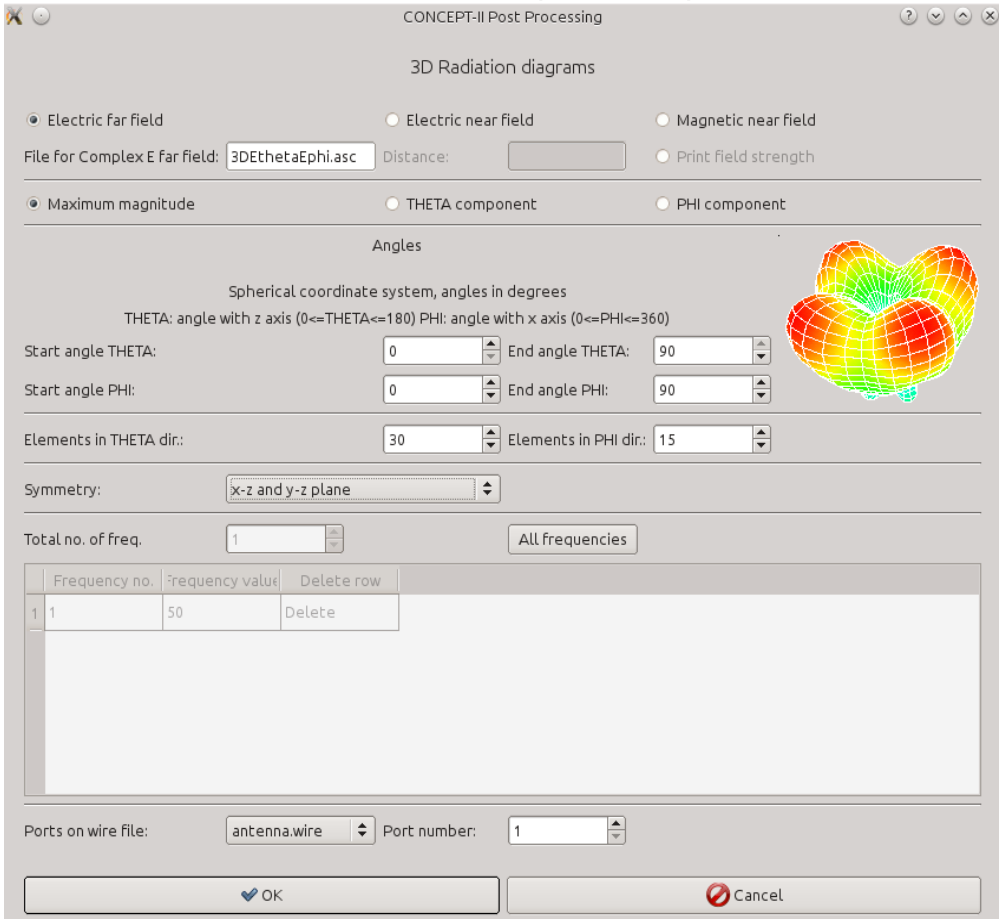


Fig 2.15: Dialogue window for creating 3D radiation diagrams

in a small sub window once the field computation is finished. Here the totally radiated power P_{rad} as derived from the electric far field values in the grid nodes can be found as well as the maximum E field strength [V] (see last chapter for explanation) and the directivity. The computations are carried out based on the following equations:

$$P_{rad}=\frac{1}{2\gamma}\iint|\hat{E}_{\Theta}|^2+|\hat{E}_{\Phi}|^2 r^2\sin\Theta\,d\Theta\,d\Phi\,,\qquad D=\frac{4\pi U}{P_{rad}}\,,\qquad U=\frac{r^2}{2\gamma}|\vec{E}(r,\Theta,\Phi)|^2$$

D: Directivity, U: radiation intensity.

Numerically a certain deviation has to be accepted between input power and radiated power. From experience is known that the following ratio should hold in case of a far-field-based analysis:

$$\frac{\text{Radiated Power}}{\text{Input Power + Dissipated Power}} \approx 0.90,\ldots,1.1$$

If this ratio is not within the given limits, the structure should be investigated carefully. In many cases the segmentation of the fed wire(s) is insufficient or the grid is bad near the generator position.

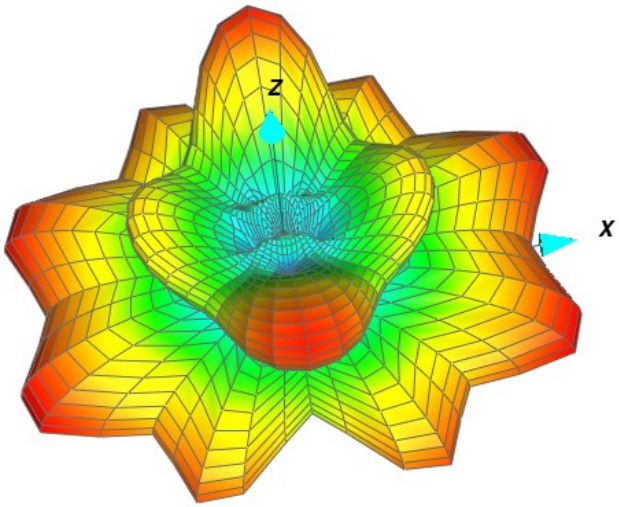


Fig 2.16: Example of a 3D radiation diagram (top-loaded antenna on a table, 150 MHz)

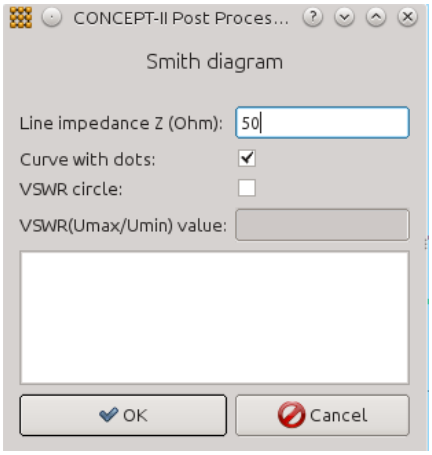
Both electric and the magnetic 3D near field distributions can be computed (no far field approximation). In such cases a “Distance” has to be entered which is simply the radius of a sphere, the center of which is located in the origin of the input coordinate system. The value under *The total radiated power* in *radiate.out* may be completely wrong if the radius does not correspond to a typical far field distance. Although a near field expression is used to compute the E field distribution, the far field formula as given above is used for P_{rad}.! An example of a 3D radiation diagram is depicted in Fig 2.16. Notice that the shape is different in the x-z and the y-z plane according to the fact that the platform which carries the antenna is closed only on two sides.The format of the *File for complex E far field* data (ASCII, default name *3DEthetaEphi.asc)* can be taken from the head section of the file.

2.1.2.6 Creating Smith diagrams

Click on ; a window as illustrated in Fig. 2.17 opens.

The characteristic impedance of the voltage generator feed transmission line has to be entered. Clicking on ‘Run smith’ automatically generates the PostScript file *smith.ps*, which is displayed in an extra window.

Fig 2.17: Window to create Smith diagrams



3 Important topics

3.1 Dielectric bodies, types of surfaces

The following surface types are available:

Surface type 'Dielectric boundary'

This is a surface separating two regions regions in space, the boundary conditions $E_{t1}=E_{t2}, H_{t1}=H_{t2}$ are enforced, leading to a double-layer current distribution. see *surf.2* in Fig 3.1 or *surf.4* and *surf.6* in Fig 3.2 as examples. Each current distributions has an impact only on one side. The number of unknowns doubles compared to a single-layer current distribution which is computed on simple metallic surfaces where only $E_t=0$ is matched (EFIE problem), see surface type 'Free space' for example.

Surface type 'Metallic boundary'

This is an ideally conducting surface separating two media. The boundary condition $E_{\tan}=0$ is matched separately on both sides leading to a double-layer current distribution as in the case of a dielectric boundary. An example is *surf.1* in Fig 3.1, note the current distributions $\vec{J}_{\text{internal}}$, $\vec{J}_{\text{external}}$.

Surface type 'Free space'

A surface in free space outside a body; an example is *surf.3* in Fig 3.1. Such a surface carries a single-layer current. The boundary condition $E_{\tan}=0$ is applied. A 'Free space' surface type can be galvanically connected to a surface of type 'Metallic boundary', which partially covers a dielectric body.

Surface types 'In body 1' (In body 2, ...)

Metallic surfaces in body 1, body 2, body 3 etc.. These surfaces can be connected to 'Metallic boundary' type surfaces and are treated by the EFIE only (single-layer current distributions) similar the the 'Free space' type; example: surface with $\vec{J}_1$ in Fig 3.1.

In the case of a surface forming the common boundary between two bodies which are attached to each other, the necessary discretization depends on the smallest internal wavelength which is smaller than the free space wavelength in general due to the internal material properties (rule: 8...10 patches per wavelength). The same is true for all surfaces forming body-internal scatterers or radiators, also wires.

Note	A dielectric body must always be **completely covered** with surfaces of type 'Metallic boundary' in conjunction with the type 'Dielectric boundary' or of type 'Dielectric boundary' alone.

These surfaces carry a double-layer current distribution, i.e., we have 2 unknowns per common edge (between two patches). Simple surfaces inside or outside bodies carry a single current layers.

3.1.1 Bodies of high conducting material

The ratio of maximum body size (typical dimension, longest extension) and skin depth should not be too large. Large conductivities are leading to a numerical instability. If such lossy structures are to be computed please refer to Section 3.1.6

3.1.2 Field-coupling through apertures

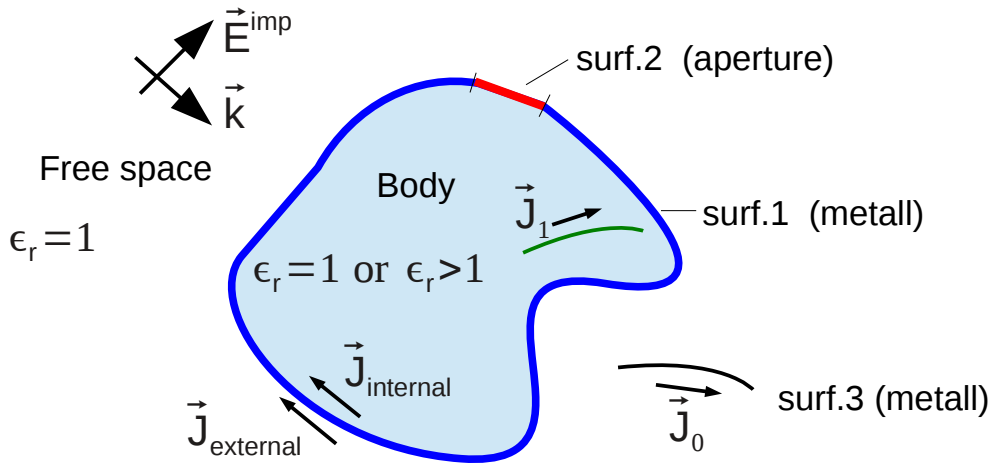


Fig 3.1: Cross section of a cavity in space. The red marked section designates an aperture (window, slot) where EM fields are able to couple in. A body must be covered by a completely closed grid.

Fig. 3.1 shows a body with an aperture, illuminated by a plane wave field.

surf.2 which covers a **small** aperture is of type 'Dielectric boundary' and *surf.1* is a metallic surface (type 'Metallic boundary'). The dielectric constant of the material inside the body can be set to one so that air is both inside and outside. For the sake of simplicity only names like surf.1, surf.2 are used, names like *aperture.surf* or *PEC.surf* could also be used.

Case: air *inside* and *outside* the body according to Fig 3.1, model with *surf.1* and *surf.2*

In principle an analysis based on *surf.1* only, i.e. solely based on the EFIE with a single-layer current distribution would be possible, but may lead to inaccurate internal fields depending on the size of the aperture – especially in case of line matching (the default matching technique). For not too small aperture sizes Galerkin matching in conjunction with a single-layer current distribution (pure EFIE solution) may already be sufficient.

It turns out that the analysis based on a completely closed grid with *surf.1* and *surf.2* on the aperture carrying a double-layer current distribution provides reliable results with respect to the coupled-in or coupled-out fields for very small apertures. This statement is especially true for bodies where internal resonances already start to appear.

Although the number of unknowns is more than twice as large compared to the case of a simple metallic cavity with aperture assuming a standard EFIE solution (only surf.1 is present, single-layer electric current distribution, line matching), the

internal field distribution can be computed much more precisely, especially between resonance peaks. Thus the body formulation with two different current layers located on *surf.1* and *surf.2* provides a powerful means for the investigation of field coupling through small apertures. Disadvantages of this method are the increased computation time and the increased modeling effort.

Details about the theoretical back ground can be found here:

A. Schröder, G. A. Rasek, H.-D. Brüns, Z. Reznicek, J. Kucera, J. E. Loos, C. Schuster: Analysis of High Intensity Radiated Field Coupling into Aircraft Using the Method of Moments. IB. 2014EEE Trans. on EMC, vol. 56, no. 1, FE

To achieve a good numerical result in the case of aperture coupling it is important that the surface elements along the aperture boundaries are smaller then the radius or width of the aperture, respectively.

3.1.3 Multi-dielectric block with symmetry

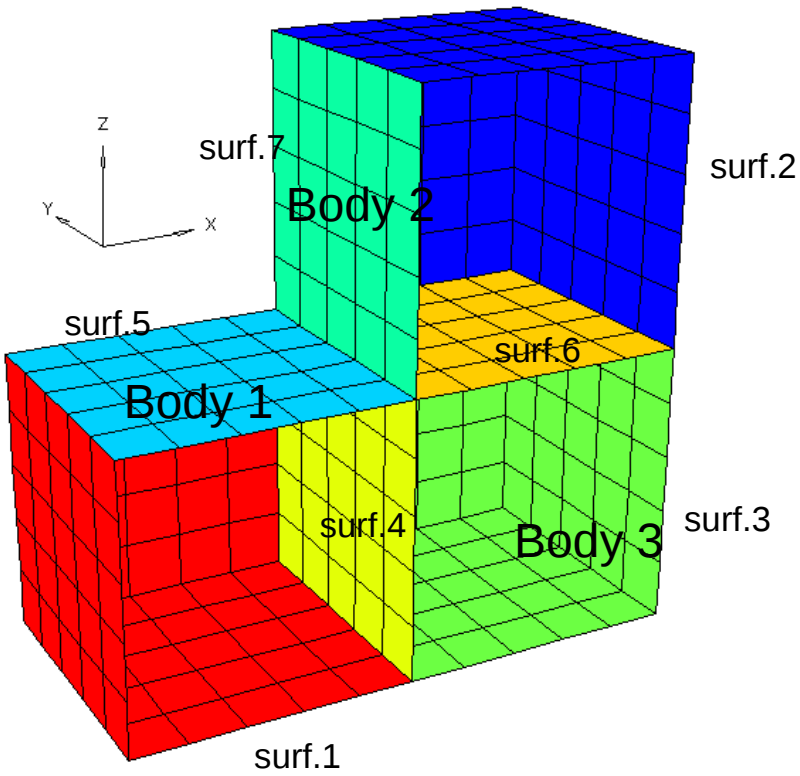


Fig 3.2: Three different dielectric blocks attached to each other

A dielectric structure (see *$CONCEPT/examples/Three_cubes*) consisting of 3 different regions (bodies, here cubical blocks) is shown in Fig. 3.2. Body 1 is attached to body 3 and body 2 is attached to body 3. The whole struture is located in free space and is illuminated by a plane wave with both the E vector and the k

vector lying in the x-z plane.

All three cubes comprise different dielectric materials: $\epsilon_{r,\text{body }1}=3;\ \epsilon_{r,\text{body }2}=4;\ \epsilon_{r,\text{body }3}=5$ and all surfaces are of the type 'Dielectric boundary'. The open front correspond to the x-z plane which is a magnetic symmetry plane. It is not allowed (and would make no sense) to place a surface of type 'Dielectric boundary' into a symmetry plane!

Only the symmetric part of the complete block is depicted and needs to be discretized (half of the structure). In summary we see seven surfaces:

body 1 consists of *surf.1*, *surf.4* and *surf.5,*

body 2 consists of *surf.2*, *surf.6* and *surf.7,*

body 3 consists of *surf.3*, *surf.4* and *surf.6*

surf.1 and *surf.5* could have been combined to a single surface. The same holds for *surf.2* and *surf.7,* hence the minimum number of surface patch file would be five here.

Surf.4 separates the bodies 1 and 3. It is important that each patch of *surf.4* has body 1 on one side and body 3 on the other side.

Important: a body must always be completely bounded by surfaces of type 'Metallic boundary' and/or by surfaces of type 'Dielectric boundary'! So one or more surfaces of the example under discussion could be of type 'Metallic boundary'. It should be avoided to cover a region completely by surfaces of type 'Metallic boundary': any amplitude of the internal current distribution would be zero because of a missing internal excitation.

3.1.4 More examples

Fig 3.3 illustrates the cross-section of structure where we have a body inside another body (body 2 in body 1) and another 2 metallic surfaces which are located in different spaces: *surf.6* is inside body 1 and *surf.5* is in free space. Free space is considered as a “special body” carrying the number 0 (body 0).

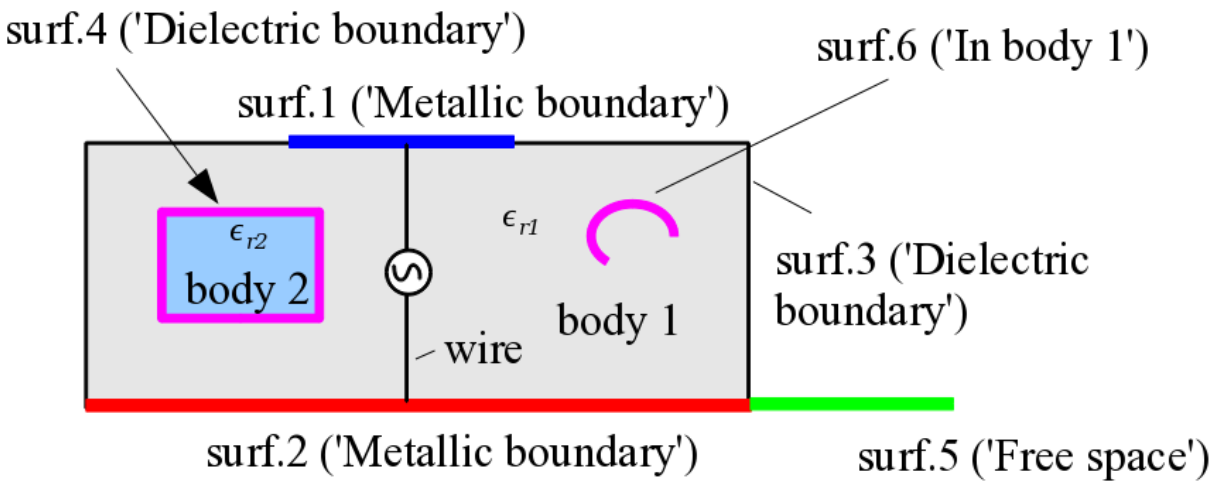


Fig 3.3: A more sophisticated example

Body 1 is bounded by *surf.1* (metal, 'Metallic boundary', blue), *surf.2* (metal, 'Metallic Boundary', red), and *surf.3* ('Dielectric boundary')

Body 2 is located completely inside body 1 and only bounded by *surf.4* ('Dielectric boundary'). The information that body 2 in not in touch with any boundary of body 1 has to be specified under the entry Body in body , see simulation project tree.

Surf.5 is a finite metallic extension of the metallic bottom plate *surf.2* and is a surface of type 'Free space'.

Surf.6 is a metallic sheet in body 1 (surface type: 'In body 1').

It is **not** necessary to specify the direction of normal vector on surfaces or patches, respectively.

A three-body problem is illustrated in Fig 3.4 (see also $CONCEPT/examples/Bodies_in_bodies). Two bodies, i.e. a cylinder and a sphere are freely floating inside a cube. The sphere is characterized by $\epsilon_r = 8$. For the cylinder we have $\epsilon_r = 4$ and for the cube we have $\epsilon_r = 2$.

Validating the solution we expect the normal components of the electric field strength to jump at each boundary by a factor of 2 ($D_{n1} = D_{n2}$).

Notice the entries in the simulation project tree and also the entries in the sub-window: The 'Cube' is in free space (default); the body 'Cylinder' is located completely inside the volume of the cube and the body 'Sphere' is completely located inside the volume of the cylinder.

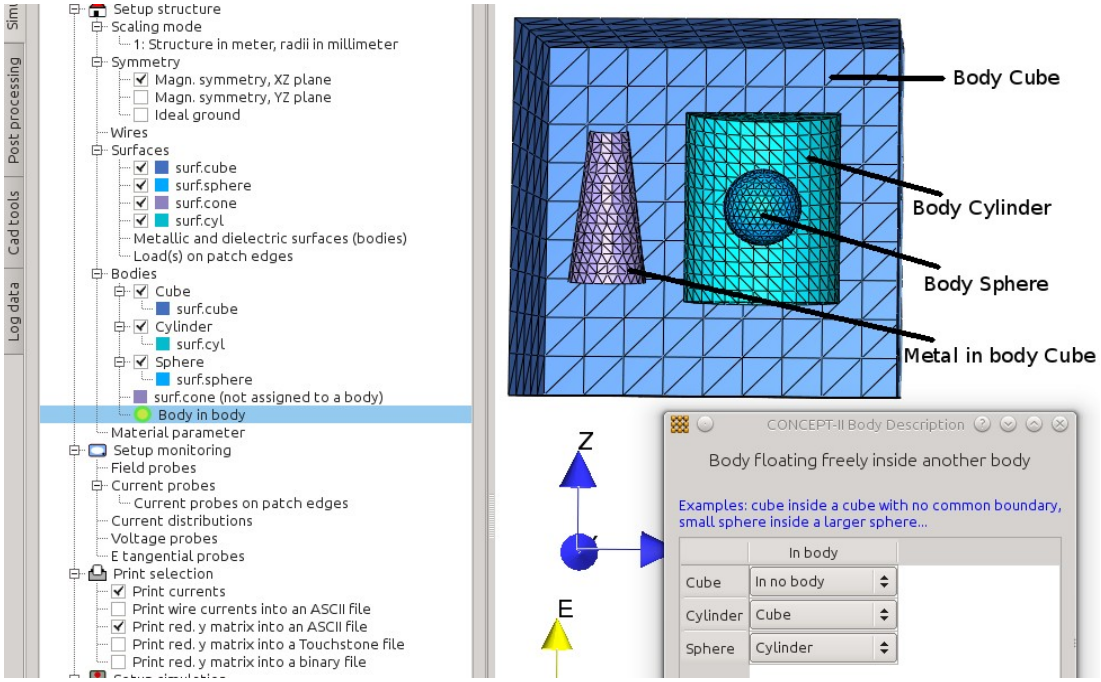


Fig 3.4: Structure according to the example under $CONCEPT/examples/Bodies_in_bodies

3.1.5 Wires in dielectric bodies, wire junctions

It is allowed to locate wires in and outside a dielectric body. It is automatically detected in which body wires are located. When a wire is attached to the node or the edge of a surface of type 'Dielectric boundary', all patches on that node are automatically “metalized”. Introduce a fine mesh at such points!

Wires can can only be attached to patch nodes.

In the case of a triangle only a single wire-node junction is allowed at this triangle. In other words: a wire or more wires can only be attached to one vertex per triangle.

Wires are not allowed to “run through surface patches somewhere”. If a thin rod is set forth into the adjacent body, the user has to enter 2 parts: one wire must end on a node and the next wire must start on the same node, but into the neighboring body.

Pay attention for a sufficient wire segmentation inside bodies. The wavelength is reduced there in case of dielectric material.

It is forbidden to place a wire directly along a dielectric boundary with zero distance or directly on a metallic surface without any height.

Note	When running wires over a discretized ground to model a transmission line the edge length of patches perpendicular to the wire axis should at be least 1,...,2 times the height of the wire. If the mesh is too rough the characteristic impedance of the wire cannot be captured numerically. This is an additional requirement to the one already explained in Fig 1.2.

If a wire is attached to a surface node with including one or more quadrangles all quadrangles are automatically turned into triangles.

It is important that the wire radius remains much smaller than the shortest length of the patch edges that connected to the node. If the ratio edge/radius is too large an error message is given and the program stops.

3.1.6 Layers on bodies

The computation of the shielding effectiveness (SE) of realistic enclosures of arbitrary geometry in a full-wave simulation is difficult for numerical methods. A reason for this are the geometrically thin conducting wall, i.e. the small distance between outer and inner surfaces where a rapid decay of the local volume currents occurs (keyword: penetration depth). This means for volume-based methods such as the finite element technique that a very fine discretization of the wall region has to be introduced which cannot be set forth all over the volume of the shielded cavity. Also in the MoM surface current simulation severe difficulties appear. Therefore a special hybrid approach has been included in *CONCEPT-II* which is able to handle cases where the wall thickness comprises several penetration depths. In this way SE values of more than 100 dB can be determined with high numerical precision.

Surfaces which are bounding bodies can be coated by thin layers, the material of which is assumed to be homogeneous and isotropic and is characterized by a conductivity, a relative permittivity ϵ_r , and a relative permeability μ_r . Practical examples are conducting foils, protective metalized clothing, or glass that has a thin metal film, vaporized onto its surface. From numerical reasons (high conductivity, see Sec. 3.1.1, very close vicinity of surface patches) such structures cannot be treated as a two-body problem with one body being the wall and another body being the cavity.

Important: Only surfaces of type 'Dielectric boundary' can be layered.

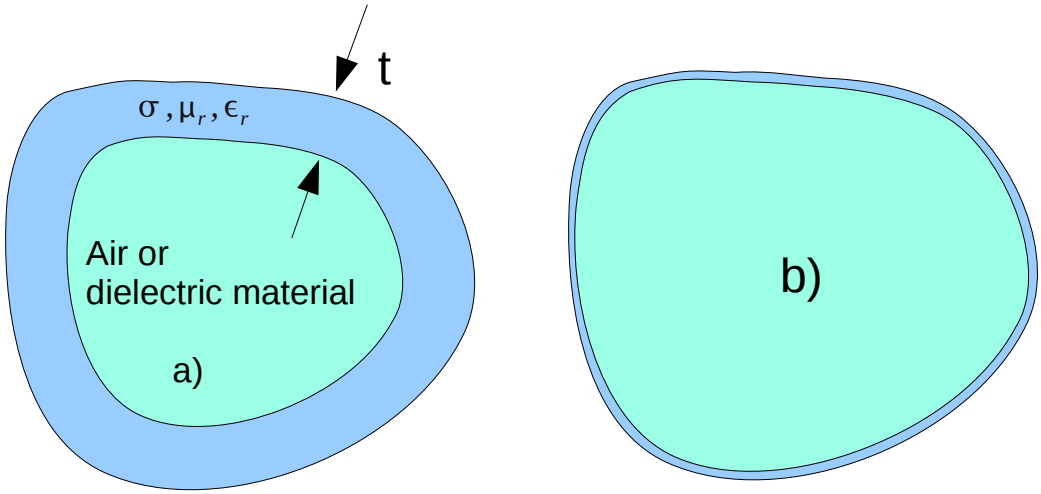


Fig 3.5: Body structures for the explanation of the layer technique. a) This problem could be treated as a two-body problem. b) In this case the wall thickness is too small to consider this as a two-body problem. Here the layer technique can be applied.

The layer technique is a suitable approach for the computation of the shielding effectiveness of enclosures which are considered as bodies filled with air ($\epsilon_r=1, \mu_r=1, \sigma=0.0$ .The approach can be applied under the following conditions:

- **The thickness of the layer has to be geometrically small**, i.e. small compared to the dimension of the body that is covered (see Fig. 3.5b). A thickness of some micrometers or millimeters on a body of typical size of 1m is not a challenge, but a “layer” of 0.5 cm thickness on a sphere with 10 cm radius marks already the limit of the applicability of the implemented hybrid technique (the MoM is combined with an analytical solution concerning the wave propagation inside the layer, see Sec. 4 for details. A structure as depicted in Fig. 3.5a) could already be treated as a multibody problem.
- **The layer should have a certain conductivity**. The implemented method works best if the wave propagation inside the layer is essentially perpendicular to the layer walls, i.e. a certain conductivity in conjunction with high values for the relative permittivity are advantageous.
- **The electrical thickness of the layer should not exceed 5,...,6 times the penetration depth** $w<(5,\cdots,6)\cdot\delta$, with $\delta=\sqrt{2/(\omega\mu\sigma)}$ and w as the geometrical thickness of the layer. Violation of this rule may lead to instable current distributions.

Fields or currents inside the layers cannot be displayed. The electromagnetic fields inside layers are analytical solutions of the one-dimensional Helmholtz equations for E and H, respectively, see also Sec. 4.

3.1.6.1 Example of a cylinder with a lossy wall of high relative permeability

A simple numerical example can be found in *$CONCEPT/examples/Cylinder_with_layer* and is illustrated in Fig. 3.6 . Only a quarter of the overall cylinder which is illuminated by a plane (1V/m), is shown. The height of the cylinder is 3.6 m and the radius is 0.3 m.

The wall of the cylinder including the top and bottom plate is characterized by $\sigma = 1824000\, S/m$, $\mu_r = 11000$ and the wall thickness 0.8 mm. The frequency is 1 kHz. Note that very high values for the relative permeability cause no difficulties.

The advantage of this example is that reference results can be computed by an analytical expression, which can be found in the famous book of Heinrich Kaden [Ka-59]. This formula is valid for infinitely long cylinders. Such a structure cannot be modeled in *CONCEPT-II*. In order to get comparable results the length of the cylinder has been chosen 6 times the diameter. For further details see the *README* file in the mentioned directory.

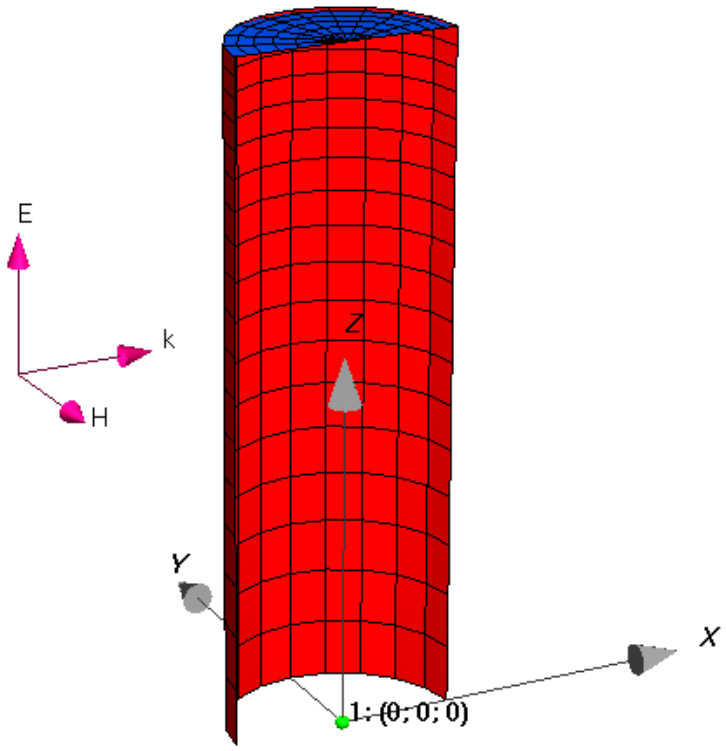


Fig 3.6: A cylinder with a lossy magnetic wall, illuminated by a plane wave field.

Symmetry has been used with respect to the x-z plane. A field observation point has been placed at (0., 0., 0.) m. Note the observation point is marked by a green dot at the center of the coordinate system in Fig. 3.6. The cylinder is modeled by

surf.1 (cylinder wall, red) and *surf.2* (top and bottom plate, blue).

It is possible to assume more than one layer on a surface. Note: In the case of multi-layers the material constants of that layer which is on the side of the body with the lowest number, here free space (0), has to be specified first.

Another area of application is the consideration of losses in metallic blocks. Imagine for example a monopole antenna mounted on a lossy metallic block. Although the block is solid and cannot be treated by the body approach (see Sec. 3.1.1), a finite wall thickness of 5...6 times the penetration depth can be assumed. So the effect of the losses on the antenna input impedance can be computed quite precisely.

[Ka-59]: H. Kaden. Wirbelstroeme und Schirmung in der Nachrichtentechnik.

Springer Verlag 1959

3.2 N-ports, S-parameters

An introduction to the topic has already been given in In Sec. 22 (item **Ports**).

To describe the interaction of ports it is not necessary to consider the system matrix. The mutual interference between ports can be described by a reduced admittance matrix of order *K*, where *K* is the number of ports, which have been defined. In general *K* is much smaller than the order N of the system matrix.

With the *CONCEPT-II* network tool (not yet included in the package, Jan. 2014) lumped loads at ports can easily be changed after the back-end run. The impact of such a change onto the other ports can be calculated very fast without solving the large MoM system of equations again as only the small admittance matrix is involved.

In the method of moments the system of linear equations can be written as follows:

$$[U]_N=([Z]_{NN}+[Z_L]_{NN})\cdot[I]_N \quad .$$

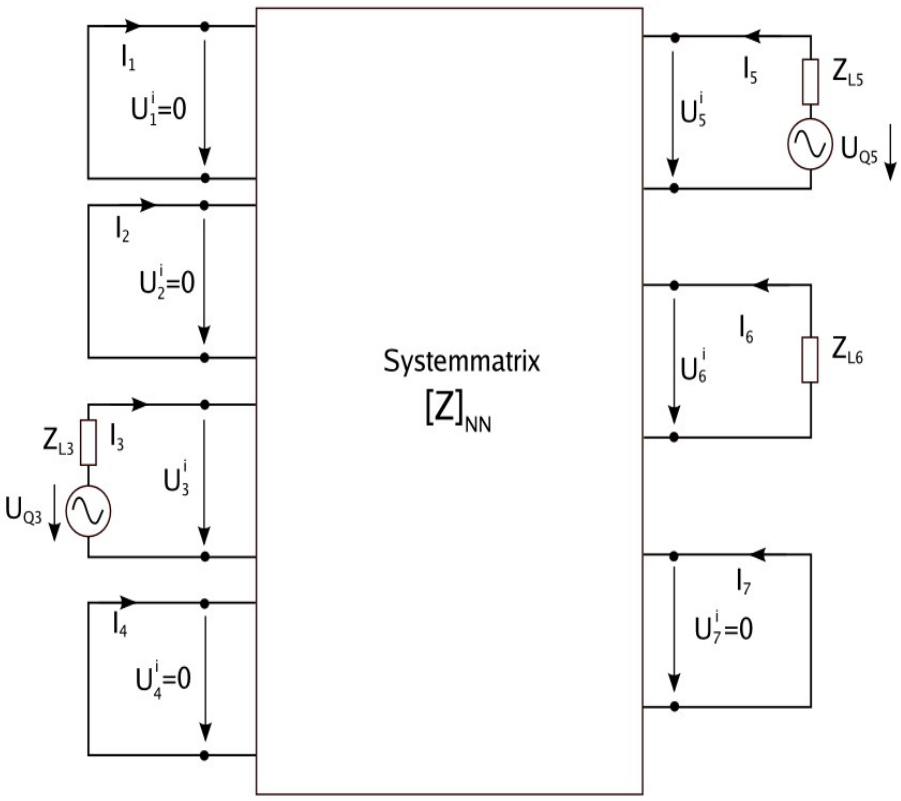


Fig 3.7: An example for demonstrating the principle. We consider a small structure with N=7. Ports are defined at positions 3 and 5

$[Z]_{NN}$ is the system matrix, $[U]_N$ is the voltage vector (excitation), $[Z_L]_{NN}$ is a diagonal matrix and contains the lumped loads (only wires can be loaded currently), all off-diagonal elements are zero, $[I]_N$ is the current vector (amplitudes of the basis functions).

Ports are defined by setting K voltage or power generators at *K* different positions. As an example consider Fig. 3.7 where a structure with N = 7 is depicted; we see generators at segments 3 and 5.

N=7 shall be reduced to *K*=2. For this purpose the equation system of order *N* has to be solved *K* times to compute *K* corresponding global current distributions, which can easily be carried out by means of the LU decomposition technique. From this data it is possible to extract the following reduced equation system:

$$[I]_K = [Y]_{KK} \cdot [U^i]_K \quad .$$

$[Y]_{KK}$ is the so-called reduced admittance matrix. From the viewpoint of the ports the impact of the whole structure, for example a ship or an aircraft is contained in $[Y]_{KK}$. The reduced admittance matrix is stored in the file *red-y-mat.asc.*

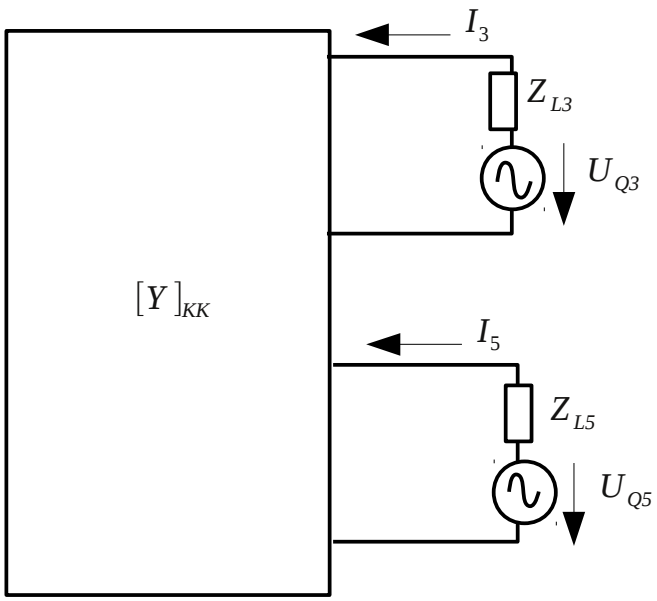


Fig 3.8: The system according to Fig. 3.7 has been reduced to a two-port system. These ports can be loaded in an arbitrary manner after the back end has finished.

S-parameters can be computed from the reduced admittance matrix by means of

$$[S]=(Y_0\cdot[I]+[Y])^{-1}\cdot(Y_0\cdot[I]-[Y])$$

with

$Y_0=1/Z_0$. $[I]$ is the identity matrix and $[Y]$ is the reduced admittance matrix as described above.

3.2.1 Files

- *red-y-matrix.asc*

Contains the reduced Y matrices for all frequency steps. Note that the Y matrices are symmetric. Each matrix comprises K * K lines with K as the number of ports.

Before storing the reduced Y-parameter matrix to the hard disk all loads are removed, so that *red-y-matrix.asc* always describes the coupling between unloaded ports (in the example K=2 is assumed)

File structure of *red-y-matrix.asc*:

Line 1: Number of ports K

Line 2: frequency

Line 3: First element of [Y]: Y(1,1) real part imaginary part

Line 4 Y (1,2) real part imaginary part

Y(2,1)

Y(2,2)

f Next frequency step

Next Y matrix, coefficients line by line

y(1,1)

y(1,2)

...

- red-y-mat.bin

Line 1: y_11 y_12 y_13 … y_1K

Line 2: y_21 y_22 y_23 … y_2K

…

Line K: y_K1 y_K2 … y_KK

Line K+1: y_11 y_12 y_13 … y_1K

Line K+2: y_21 y_22 y_23 … y_2K

…

Line K+K: y_K1 y_K2 … y_KK

…

(for all frequency steps)

Y_nk : real part imagniary part, double precision each

- red-y-mat.s(n)p

The same as red-y-mat.asc but written in Touchstone II format. A Touchstone file is an ASCII text file that is used for documenting n-port network parameter data . This format has become a de facto industry-standard file format, see http://en.wikipedia.org/wiki/Touchstone_file for example.

.

- port-zin(*n*).asc

n is the port number. Input impedance of the *n*th port. Each line contains the frequency and the real and the imaginary part of the input impedance.

3.2.2 Computation of S-parameters

Click on the button {s}, **Results Section**, main window. Values are only available if ports were defined in the front-end run.

The S-parameters are gained from the reduced Y matrix, port loads which might have been set in the original data have been erased.

Select the port numbers and the impedance “Z (Ohm)” first, see Fig. 3.9.

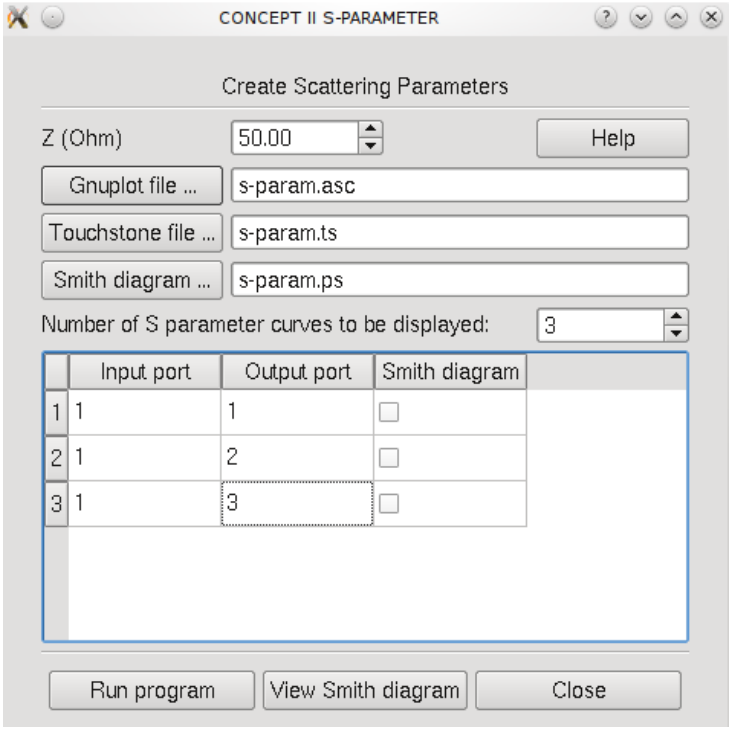


Fig 3.9: Window for the specification of S parameters to be displayed as a function of frequency

In the example (Fig 3.9) the magnitude of the following complex S parameters will be displayed as a function of frequency after a click on **Run program** : S_{11}, S_{12}, S_{33} . All data is written into the ASCII file *s-param.asc,* ready to be displayed by the *CONCEPT-II* gnuplot front end.

For displaying the real and imag. parts click on the **Scattering parameters** entry in the **Post processing** tree → S**how results**. The *CONCEPT-II* gnuplot front end opens.

Files that available in the working directory:

s-mat.asc

This file contains the S parameter matrices for all frequency steps. The format is almost identical with that of *red-y-mat.asc*. The only exception is that the first line

contains the first frequency step and not the number of ports, then follows line-by-line the S matrix for the first frequency etc..

Note again: Loads which may have been set at ports are not contained in the reduced Y matrix any more.

s-params.asc:	Default output file name, contains ASCII data. The format is explained in the head section.
s-params.s2p	ASCII data file, according to the Touchstone® file format specification. All S parameters (not only the selected ones)
are	written into this file.
s-param.ps	PostScript graphics file, Smith chart; select one or more S parameters to be displayed in a single diagram.

3.2.3 EM fields when port excitations are modified

In practical cases it can be of interest computing electromagnetic fields in observation points due to changes of port voltages. By means of the port fields tool new field strength values can be quickly computed without the necessity of integrating again. Let us consider an example referring to the electric field strength. We have 4 observation points and 3 ports. This leads to a coupling matrix with has 4 rows and 3 columns. More generally we would have a matrix with

- the number of rows according to the number of field points and
- the number of columns according to the number of ports.

The following equation holds, all quantities are complex values:

$$\begin{pmatrix} E_u(\vec{r}_1) \\ E_u(\vec{r}_2) \\ E_u(\vec{r}_3) \\ E_u(\vec{r}_4) \end{pmatrix} = \begin{bmatrix} SE_{11} & SE_{12} & SE_{13} \\ SE_{21} & SE_{22} & SE_{23} \\ SE_{31} & SE_{32} & SE_{33} \\ SE_{41} & SE_{42} & SE_{43} \end{bmatrix}_u \cdot \begin{pmatrix} U_{p1} \\ U_{p2} \\ U_{p3} \end{pmatrix} = [\mathbf{SE}]_u \cdot \begin{pmatrix} U_{p1} \\ U_{p2} \\ U_{p3} \end{pmatrix}$$

$$u = x, y, z$$

In case of the H field the matrices are designated $[\mathbf{SH}]_u$. In fact three matrices are computed in each case and stored, namely $[\mathbf{SH}]_x$, $[\mathbf{SH}]_y$, $[\mathbf{SH}]_z$ for the H field and $[\mathbf{SE}]_x$, $[\mathbf{SE}]_y$, $[\mathbf{SE}]_z$ for the E field.

$\vec{E}(\vec{r}_j)$ is the E field in observation point i

U_{pj} is the (new) voltage of the port *j*. In case of only a single generator being active all remaining entries in the voltage vector can be set to zero.

Once field points have been specified and provided that port excitation (Ports, page Fehler: Referenz nicht gefunden) has been chosen and the back end run has finished, the $[\mathbf{SE}]_u$ and $[\mathbf{SH}]_u$ matrices can be set up.

Clicking on provides:

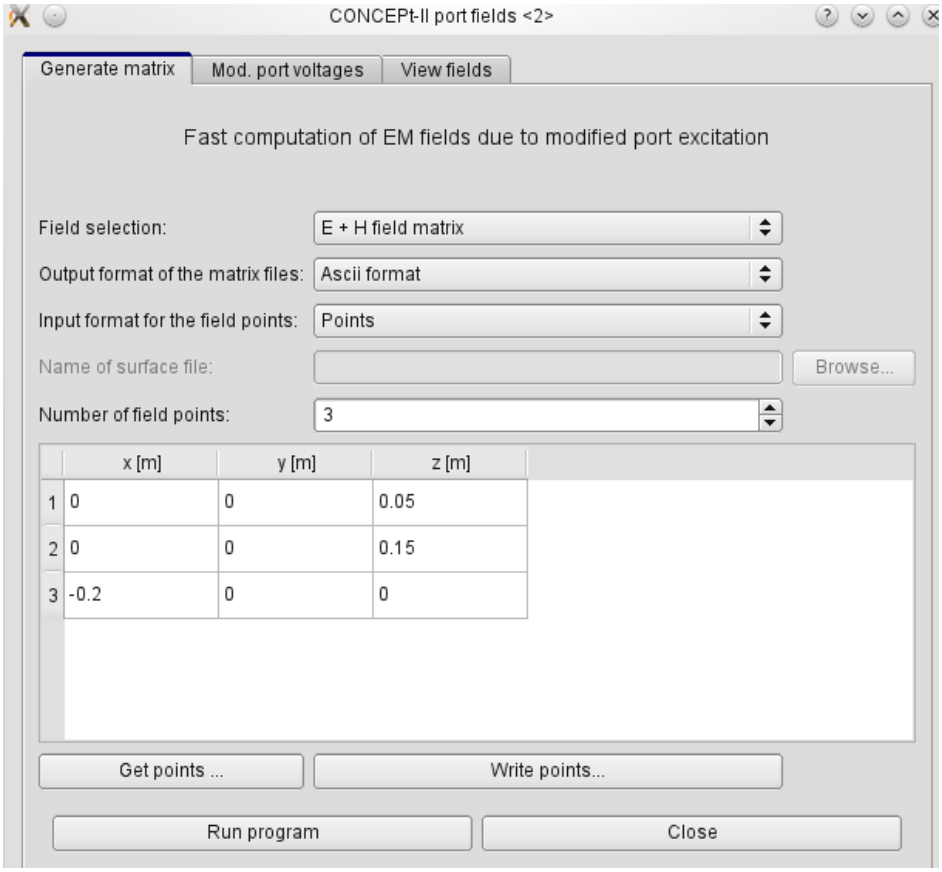


Fig 3.10: The cards designate the steps which are necessary to set up the coupling matrixes [**SE**] and [**SH**]

Step 1: Generate matrix

Step 2: Mod. port voltages

Step 3: View fields

Notice that the steps have to be carried one after another.

Example directories for exploring the tool are:

$CONCEPT/examples/3-coupled-dipoles,

Entries under the 'Generate matrix' card:

Field selection →:

- E field matrices $[\mathbf{SE}]_u$ and/or H field matrices $[\mathbf{SH}]_u$ shall be computed. Each matrix will be stored onto the hard disk in the current working directory.

  File names:

  $[\mathbf{SE}]_x$ → *sexu-matrix.asc* (for the x components of the E field)

  $[\mathbf{SE}]_y$ → *seyu-matrix.asc* (for the y component of the E field)

  $[\mathbf{SE}]_z$ → *sezu-matrix.asc* (for the z component of the E field)

  $[\mathbf{SH}]_x$ → *shxu-matrix.asc* (for the x components of the H field)

  $[\mathbf{SH}]_y$ → *shyu-matrix.asc* (for the y component of the H field)

  $[\mathbf{SH}]_z$ → *shzu-matrix.asc* (for the z component of the H field)

  The file format is identical for all sub matrices (example: $[\mathbf{SE}]_x$, 3 field points and 2 ports → 3 rows, 2 columns for each matrix for each frequency). Entries row by row.

  Index r: real part, index i: imag. part

  np: number of field points, npo: number of ports

  First frequency (first row)

  Exr(1,1) Exi(1,1) Exr(1,2) Exi(1,2) … Exr(1,npo) Exi(1,npo) (second row)

  Exr(2,1) Exi(2,1) Exr(2,2) Exi(2,2) … Exr(2,npo) Exi(2,npo) (third row)

  Exr(3,1) Exi(3,1) Exr(3,2) Exi(3,2) … Exr(3,npo) Exi(3,npo)

  …

  Exr(np,1) Exi(np,1) Exr(np,2) Exi(np,2) … Exr(np,npo) Exi(np,npo)

  Second frequency

  Exr(1,1) Exi(1,1) Exr(1,2) Exi(1,2) … Exr(1,npo) Exi(1,npo)

  Exr(2,1) Exi(2,1) Exr(2,2) Exi(2,2) … Exr(2,npo) Exi(2,npo)

  Exr(3,1) Exi(3,1) Exr(3,2) Exi(3,2) … Exr(3,npo) Exi(3,npo)

  Exr(np,1) Exi(np,1) Exr(np,2) Exi(np,2) … Exr(np,npo) Exi(np,npo)

  …

  nth frequency

  Exr(1,1) Exi(1,1) Exr(1,2) Exi(1,2) … Exr(1,npo) Exi(1,npo)

  Exr(2,1) Exi(2,1) Exr(2,2) Exi(2,2) … Exr(2,npo) Exi(2,npo)

  Exr(3,1) Exi(3,1) Exr(3,2) Exi(3,2) … Exr(3,npo) Exi(3,npo) ⋯

Exr(np,1) Exi(np,1) Exr(np,2) Exi(np,2) … Exr(np,npo) Exi(np,npo)

Output format of the matrix files →

- ASCII
- Binary

  In this case the suffixes of the matrix files are changed to .bin, for example *sexu-matrix.asc* is turned to *sexu-matrix.bin*

Input format of the field points →

- Points

  In this case the number of field points can be entered directly as demonstrated in Fig. 3.10 for 3 field points or can be read from a file of arbitrary name, default name is

  EH-coord.asc

  File format of *EH-coord.asc*:

  x_1 y_1 z_1 (coordinates of the 1. point)
  x_2 y_2 z_2 (coordinates of the 2. point)
  x_3 y_3 z_3
  …
  x_np y_np z_np np: number of field points (np*th* row)

- Surface file, field points in the center of each patch

  CONCEPT-II surface patch file format. The surface can be of arbitrary shape. The field distribution on the surface can be visualized later under the card 'View fields' (in case of a frequency loop as a movie).

Clicking on **Run program** sets up the mentioned matrices. Depending on the number of field points and ports this may take a moment. Once all $[\mathbf{SE}]_u$, $[\mathbf{SH}]_u$ matrices are known, subsequent field computations due to modified port voltages can quickly be carried out.

The tool for generating the matrices is available at the prompt, usage:

> portfields arg1 arg2

arg1: 1 → E field, 2 → H field, 3 → both E and H field

arg 2: 1 → ASCII format, 2 → binary format

The program will automatically read the coordinates of the field points from the file *EH-coord.asc* (number of rows corresponds to the number of points, see above) and set up the described matrices $[\mathbf{SE}]_u$ and/or $[\mathbf{SH}]_u$, u=x,y,z.

When dielectric bodies are present then the ASCII file *fieldpoint-matpar.asc* will be generated. For each field point the following data is listed in a row:

Coordinates x,y,z; body no.; conductivity; rel. diel. const.; rel. magn. constant

---

Clicking on the **Mod. port voltages** card allows to change the feed voltage at each port. Note that the $[\mathbf{SE}]_u$ and $[\mathbf{SH}]_u$ matrices that were filled based on the entries of the **Generate matrix** card are taken for this purpose. It is recommended to go to the cards in a sequence:

Generate matrix → Mod. port voltages → View fields

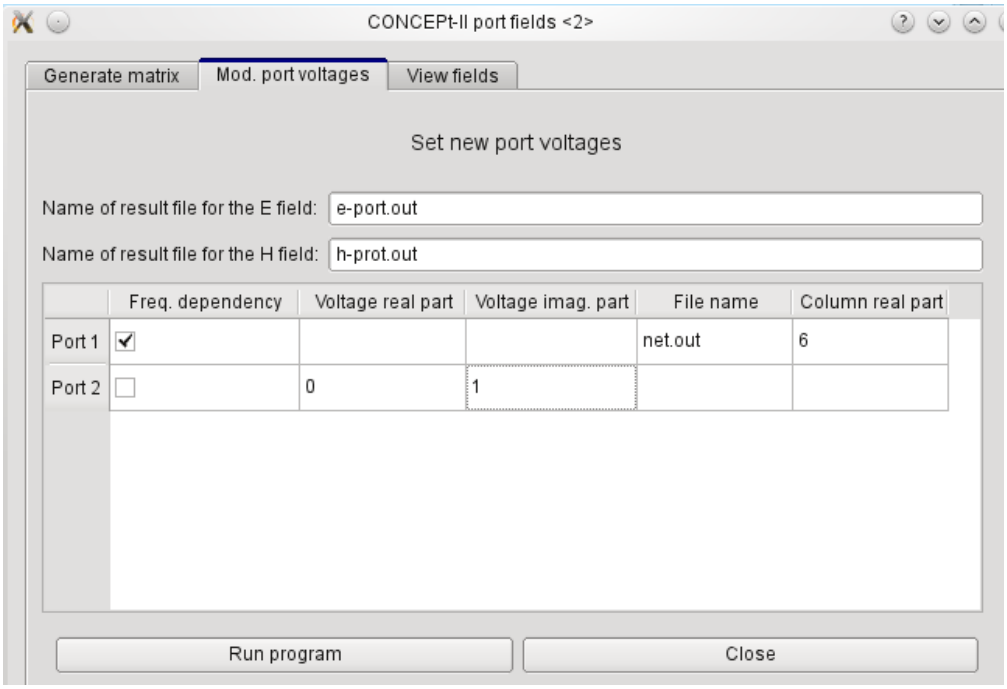


Fig 3.11: The new port voltages can be specified here

Name of the result File for the E field (H field)

The field data is written into files, the names of which can be arbitrarily chosen. In case of field points (not surface!) we have the following format (n: last frequency, np: number of field points):

###

File generated from the CONCEPT II program *modportfields*
Column 1 : frequency
Column 2 : real part x, point1
Column 3 : imag part x, point1
Column 4 : real part y, point1
Column 5 : imag part y, point1
Column 6 : real part z, point1
Column 7: imag part z, point1
⋯ 6 columns for each field point (max. 601 columns are indicated)

f1 Exr_1 Exi_1 Eyr_1 Eyi_1 Ezr_1 Ezi_1 ⋯ Eyr_np Eyi_np Ezr_np Ezi_np

f2 Exr_1 Exi_1 Eyr_1 Eyi_1 Ezr_1 Ezi_1 ⋯ Eyr_np Eyi_np Ezr_np Ezi_np

⋯

f_n Exr_1 Exi_1 Eyr_1 Eyi_1 Ezr_1 Ezi_1 ⋯ Eyr_np Eyi_np Ezr_np Ezi_np

Each field point comprises 6 columns $E_{xr}, E_{xi}, E_{yr}, E_{yi}, E_{zr}, E_{zi}$. Representation of columns as a function of frequency can be accomplished by means of the CONCEP-II gnuplot front end .

If *Freq. dependency* is active, then the port voltages are taken from a file, the name of which has to be entered in the corresponding location of *File name*. Only the column of the real part of has to be indicated. It is assumed that the column of the imaginary part is the next one.

Port voltages as a function of frequency may be computed by the network tool

(*menu bar*). The network tool provides the ASCII file *net.out* where the port voltages are stored.

If a complex port voltage does not change with frequency *Freq. dependency* should not be active (setting both the real part and the imaginary part of a port voltage to zero is allowed). If a port is not active an entry is not necessary in the corresponding row and the program automatically sets the port voltage to zero.

Clicking on the **View fields** card the user can choose what type of EM field he wants to be presented. In the case of field points (see Fig. 3.10, *Input format for the field points* → **Points**) the *CONCEPT-II* Gnuplot front end will automatically be invoked. Only the first 50 columns will be listed in the window. In the head of the corresponding data file the position of the first 601 columns will be listed.

If *Input format for the field points* → **Surface file, field points in the center of**

each patch has been selected, then the distribution of the magnitude of the field will be displayed as soon as the field computation was finished. The format of the result files has changed accordingly. Clicking on the **Animate** button the field distribution will be displayed as a function of frequency.

3.3 RCS

The radar cross-section (RCS) is important in radar engineering. The RCS is a measure of how detectable an object is when illuminated with a radar. "The RCS of a radar target is the hypothetical area required to intercept the transmitted power density at the target such that if the total intercepted power were re-radiated isotropically, the power density actually observed at the receiver is produced" (http://en.wikipedia.org -> radar cross-section). The radar cross-section is the "apparent size" of an object as seen by the radar.

Generally one has to distinguish between the so-called **bistatic** and the **monostatic** radar cross section. Currently only the monostatic case can be computed in *CONCEPT-II* (March 13)

In the case of the monostatic RCS receiver and transmitter are always located at the same position in space while moving around the radar target (orbit plane). In the bistatic case the source of excitation is fixed in space whereas the receiver rotates around the target on a circular path.

Any RCS calculation is restricted to an excitation of the radar target by a plane wave of linear polarization $\vec{E}_{inc}, \vec{H}_{inc}, \vec{k}_{inc}$. These vectors are moved with fixed mutual position (all angle 90 degrees) on the orbit plane (index *inc* for "incoming")

The calculated RCS values σ are given in $dB\lambda^2$ or $dB\,m^2$. The definition is as follows (index *scat* for "scattered"):

$$\sigma_{rcs} = \underbrace{\lim}_{R\to\infty} 4\pi R^2 \frac{|E_{scat}|^2}{|E_{inc}|^2} = \sigma_{rcs} = \underbrace{\lim}_{R\to\infty} 4\pi \frac{|R\,E_{scat}|^2}{|E_{inc}|^2}$$

$$\sigma_{dB\lambda^2} = 10\log_{10}\frac{\sigma_{rcs}}{\lambda^2}, \qquad \sigma^2_{dBm} = 10\log_{10}\frac{\sigma_{rcs}}{m^2}$$

Computing the RCS of an object includes a special kind of excitation. Thus the window according to Fig 3.12 appears under the **Simulation** tab.

Excitation → (right click) **RCS .**

Currently not possible: Ideal ground, existence of wires (June 2013).

Under treatment of RCS excitation the following items are available:

- Conventional

Treatment of multiple right-hand sides by LU-decomposition, multiple processor usage is possible.

- SVD

  Speed-up of the solution by compressing the excitation matrix; this matrix is formed by all right-hand sides. Applicable only if the number of unknowns is larger than the number of right-hand sides, only for single-processor usage, all solvers, June 13) .

- ACA

  Speed-up of the solution by compressing the excitation matrix. Applicable only if the number of unknowns is larger than the number of right-hand sides, a fraction of the coefficients forming the matrix of the right-hand side has to be computed explicitly. Only for single-processor usage, all solvers.

Accuracy of compression: Default is 1.e-5. Is a measure for the matrix approximation (all right-hand sides are forming a matrix).

Polarization: The vector of the electrical field strength is rotated around the k vector by the angle to be entered (right-hand rule applies).

Details concerning the SVD (Singular Value Decomposition) or ACA (Adaptive Cross Approximation) usage in the context of RCS computations can be found in the following paper:

A. Schröder, Heinz-D. Brüns, C. Schuster: “A Hybrid Approach for Rapid Computation of Two-dimensional Monostatic Radar Cross Section Problems With the Multilevel Fast Multipole Algorithm”, IEEE AP, vol. 60, no.12, December 2012

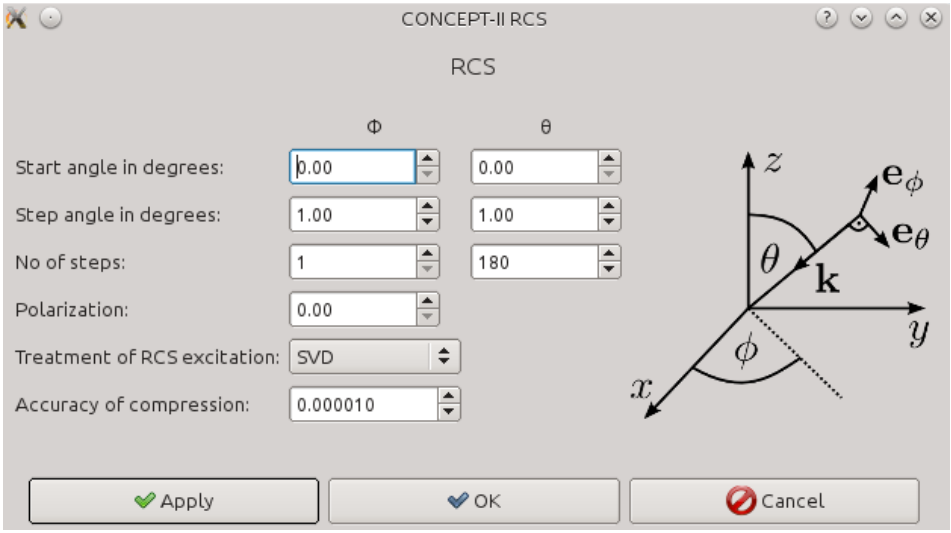


Fig 3.12: Input fields for an RCS computation (**Simulation** P*roject tree*, **Excitation → RCS**)

Note: In case of 'Conventional' a very long solution time has to be expected in conjunction with MLFMA. Hence use MLFMA only with SVD or ACA! For very large problems ACA is faster then SVD and should be preferred. The RCS can be

computed in $\Delta\Phi$ and $\Delta\Theta$ regions and along orbits around the object. The angle definition is similar to the one as described under **Plane wave field** of Sec. 1.9.2. .

Click on (**Post-processing**) → the *CONCEPT-II* window for plotting RCS

results opens (Fig 3.13); both curves and 2D representations are possible. Graphical output is possible in the *Display area* and in **Gnuplot View**, **SVG**, **PNG**, **PostScript (eps)**, see *Output to.* The RCS data can be represented in $dB\lambda^2$ or in $dB\,m^2$ (see definitions above).

An example can be found under *$CONCEPT/examples/NASA_almond*

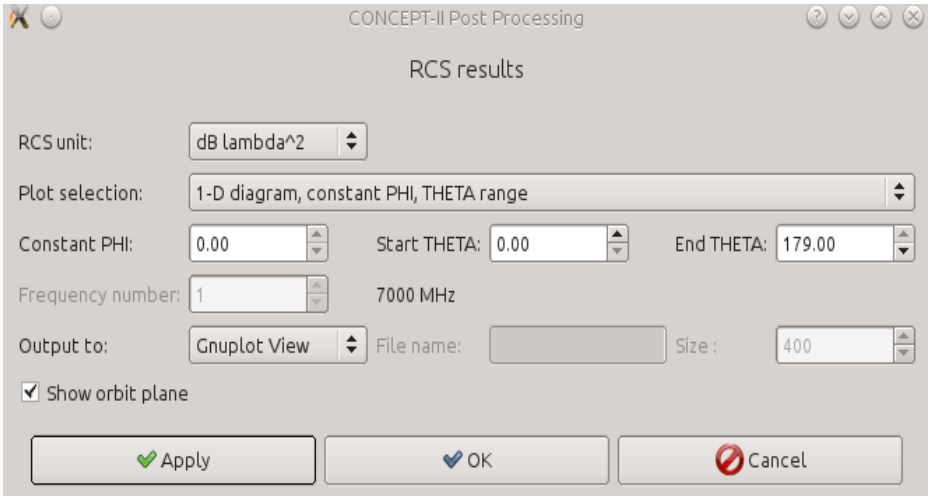


Fig 3.13: Window for plotting RCS curves (**Post processing**)

Currently it is only possible to consider objects in free space (March 13).

3.4 Computation of line integral of E and H

The following line integrals of E or H can be numerically computed:

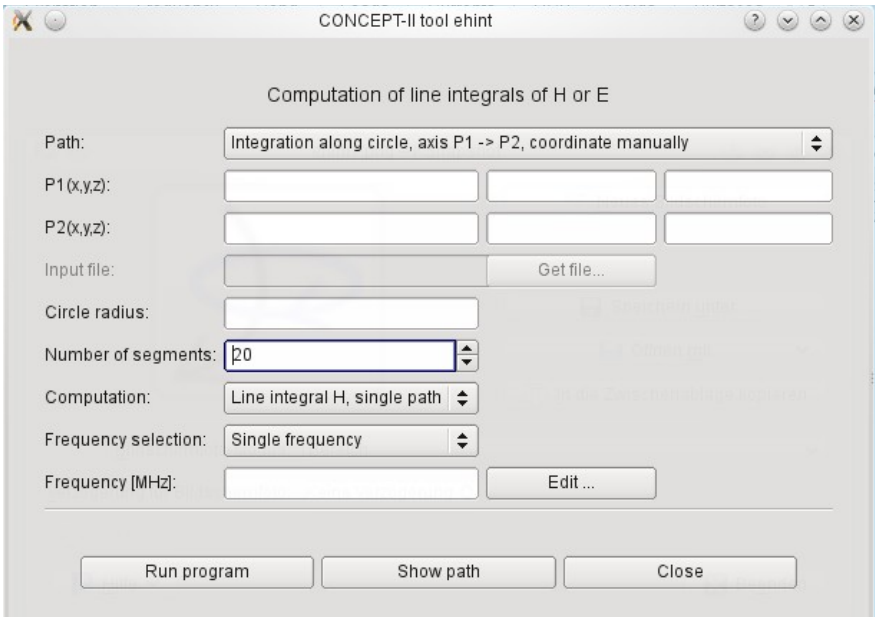


Fig 3.14: Window for the specification of paths for line integrals of E and H

$$\oint_{\partial A} \vec{E}\cdot d\vec{s} = -j\omega \iint_{A} \vec{B}\cdot d\vec{A} \quad , \quad \oint_{\partial A} \vec{H}\, d\vec{s} = I_{Wire} + \iint_{A} \left(j\omega\epsilon\vec{E} + \vec{J}_{Surface}\right)\cdot d\vec{A}$$

Clicking on opens a window according to Fig.3.14.

Path

1. **Integration on a circle, axis P1 -> P2, coordinates manually**

   For the axis definition, see Fig. 3.15.

2. **Integration on a circle, axis P1 -> P2, coordinates from file**

   P1 and P2 (see 3.15.) of the axis are taken from a file:

   Line 1 Xstart Ystart Zstart

   Line 2 Xend Ystart Zstart

   A convenient way to define the points P1 and P2 Fig 3.15 is to open the structure under **Cad tools** → **Cad tools 1** → activate *points into file* → the "Create Points" window opens. Right click on two nodes at the center distance of which the line integral shall be computed. The chosen nodes mark the ends of the axis according to Fig 3.15. Save the node coordinates into a file.

   Circle radius and the *Number of segments* (integration intervals) have to be specified

3. **Integration path from file (wire format)**

   The file must have the *CONCEPT-II* wire format: the first line contains the

number of straight sections. Each following line has the following entries.

Xstart Ystart Zstart Xend Yend Zend *Dummy* *Number of integration points*

Dummy has no meaning here, 0 is suggested.

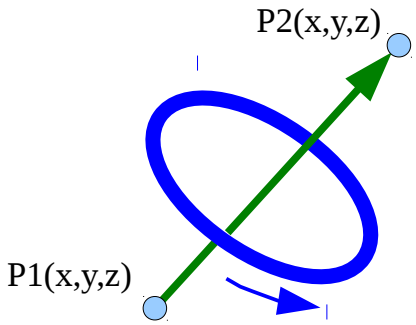


Fig 3.15: Direction of the line integration and axis of the circle (the right-hand rule applies)

The following options are available under “Computation”:

- Line integral of the E field
- Line integral of the H field-coupling
- Under the condition that **Integration path from wire file (wire format)** has been selected, voltages as a result of integrating the E field along straight paths can be computed.

  Notice that we have a behavior different from static fields. In the high frequency case the integration result depends on the shape of the path. The lowest frequency which can be computed depends on the type and geometrical size of the structure and the discretization and may vary between 100 Hz (large structure) and 10 kHz (dielectric structure).

Frequency selection: selecting **All frequencies**

The results are written into the ASCII file *EH_int_result.asc.* In case of a frequency loop the *CONCEPT-II* Gnuplot front end can be used to display curves as a function of frequency. The result of the first 4 frequency steps are shown on the screen in a special window. The corresponding file is: *EH_int_screen.asc*.

3.4.1 Example: Line integral to measure the surface current

Fig. 3.16 shows a torus, illuminated by a plane wave field (E = 1V/m) .

Torus: Radius of center line: 0.3 m
Radius of conductor (tube) cross section: 1 cm
Frequency 100 MHz

The axis of the circular integration path can be determined as described in the previous chapter, notice the blue dots. It is interesting that the surface current distribution looks rather complicated, see the detailed view in Fig. 3.17. Here we can see that the center of the integration circle does not correspond to the axis of the torus. The thin-wire assumptions are valid for the structure and therefore the torus can be approximated by a sequence of straight wire sections. One finds that the value of the "measured " current and the line current of the simple wire model are in a very good agreement. A wire circle can easily be created by means of the helix tool (**Cad tools** → **Cad tools 2**, set "Pitch angle of helix in degrees" to 0

$\text{Radius}\,(\phi_{\min})=0.3,\ \text{Radius}\,(\phi_{\max})=0.3,\ 0°,360°$).

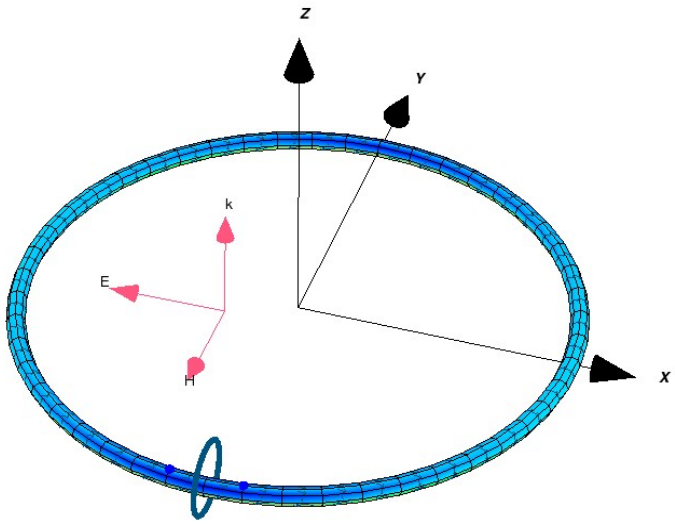


Fig 3.16: Metallic torus in a plane wave field.

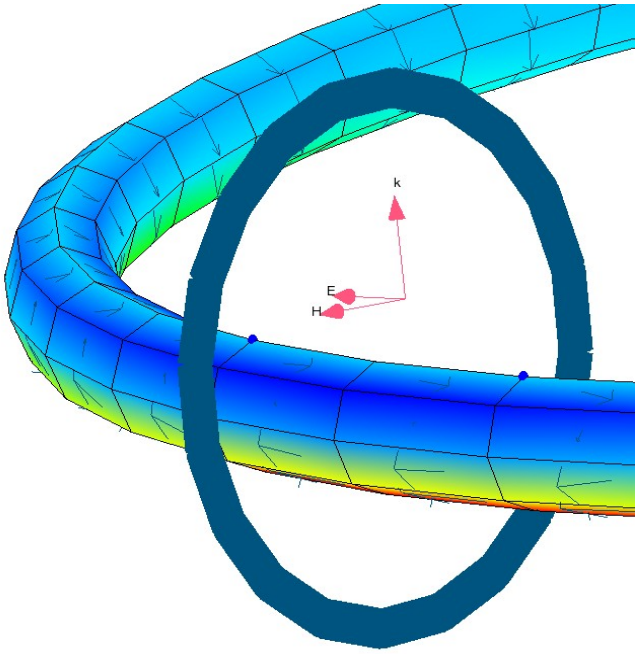


Fig 3.17: The induced surface current distribution in the region of the torus according to Fig. 3.16 where the line integral is performed. The vertical ring indicates the path of integration.

See *$CONCEPT/examples/Torus_plane_wave*

3.5 Integrating the Poynting vector on a triangulated surface

Based on the following expression the power flow through arbitrary surfaces can be computed:

$$P = \frac{1}{2}\mathrm{Re}\iint_A \hat{\vec{E}} \times \hat{\vec{H}}^* \cdot \mathrm{d}\vec{A}' \tag{1}$$

Clicking on the window according to Fig. 3.18 opens.

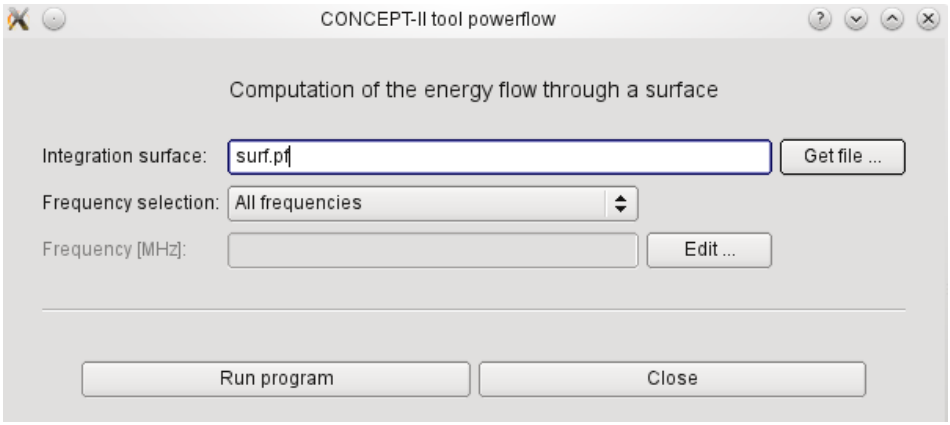


Fig 3.18: The input window for the computation of the power flow

Integration surface

Here the integration surface (Eq. (1)) has to be specified. Required is the *CONCEPT-II* patch format and only triangles are allowed. Creating such an integration surface is possible by means of tools, which available under **Cad tools**, for example . The orientation of the normal vectors can easily be checked by clicking on .

Example: completely closed surface, all normals are pointing to the outside, an antenna is inside the integration surface → the resulting power value will be positive. If power is flowing from the exterior to the exterior, the power result will be negative (scattering case). The power flow density distribution is displayed on the integration surface.

Frequency selection

If **All frequency steps** has been selected the result of the first 4 steps is displayed on the screen (special result file: *PowerScreen.asc*)

Result file: *PowerResult.asc* → ready to be displayed using .

3.5.1 Example: Loaded circular wire loop

Let us consider the same loop as shown in Fig. 3.16 as a pure wire loop. Here the loop is turned around the x axis by 90 degrees, the loop area lies in the x-z plane now. The loop is loaded by a resistor of 50 Ω at the center of the first wire which is beginning at z=0 m. Thirty segments are forming the circumference (R_{loop} = 0.3 m, R_{wires} = 5 mm, f = 100 MHz). The loop is illuminated by a plane wave field (1V/m) such that the wave is traveling in positive z direction and the H vector is perpendicularly oriented to the loop area as illustrated in Fig. 3.12.

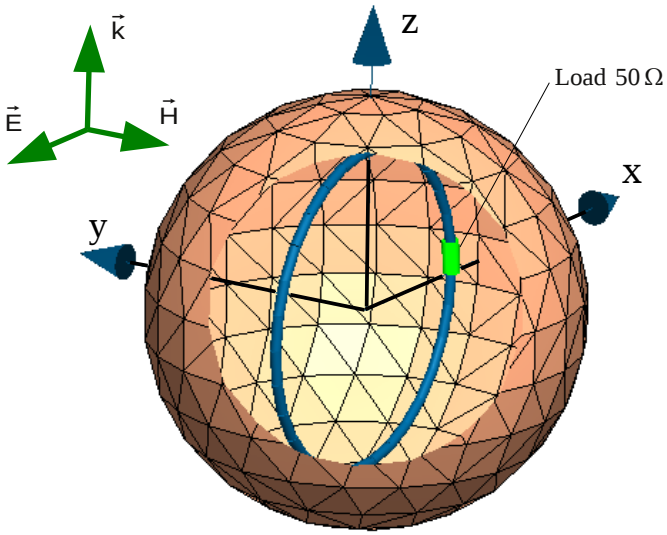


Fig 3.19: Circular loop loaded by a resistor in a plane wave field. The wire loop is inside a spherical integration surface

The patches of the integration surface should not get too close to the structure. At least 8 triangular patches per wavelength are necessary in case of an electrically large circumference (default surface name: *surf.pf*). In case of a short wavelength enough patches have to be set to approximate the surface of the sphere properly. There is no need to locate the integration surface in the far field.

This example is under *$CONCEPT/examples/Torus_plane_wave*

The power which is dissipated in the load can be taken from *concept.out*, provided that the print option has been switched on and a voltage probe has been specified. This value has to be compared with the result of the power flow tool.

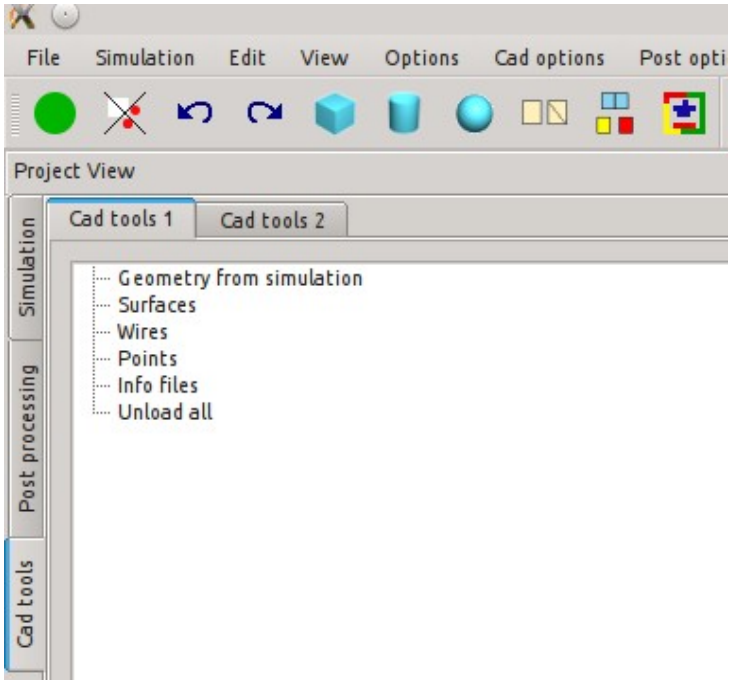


Fig 3.20: The items of the CAD Tools tree

3.6 Discretizing structures

Wires and surfaces can be meshed by activating the **CAD tools** tab → the corresponding tree appears in the 'Project view' section as depicted in Fig 3.20)

Geometry from simulation → **Load simulation files:** Take over all wire and surfaces files as specified under **Simulation**

Surfaces, **Wires**, **Points** → load or unload files with corresponding data

: deselect a selection of patches or nodes

: Turn a quadrangular mesh into a pure triangular mesh

: The “explode” tool. It decomposes a global surface into sub-surfaces. Details are described in the head section of the window.

: Running the explode tool on large structures that have been converted from external tools into the *CONCEPT-II* surface patch data format a lot of files might appear. By this tool sub surfaces can be quickly unloaded by mouse selection. The corresponding file names can be written into a list. This list can be used to create a single surface with a unique patch and node numbering by means of the **combi**ne

tool (**Cad tools 2**,)

3.6.1 Spheres, plates, cones, cylinders etc.

For a quick generation of various simple surfaces and bodies three tools are available, the icons of which can be seen in the menu bar . Each of these tools provides the possibility of a frequency dependent discretization. Be careful with electrically small structures – in such cases the provided grids might be insufficient. Simply enhance the frequency or the number of patches per wavelength for a proper grid in the case of electrically small structures.

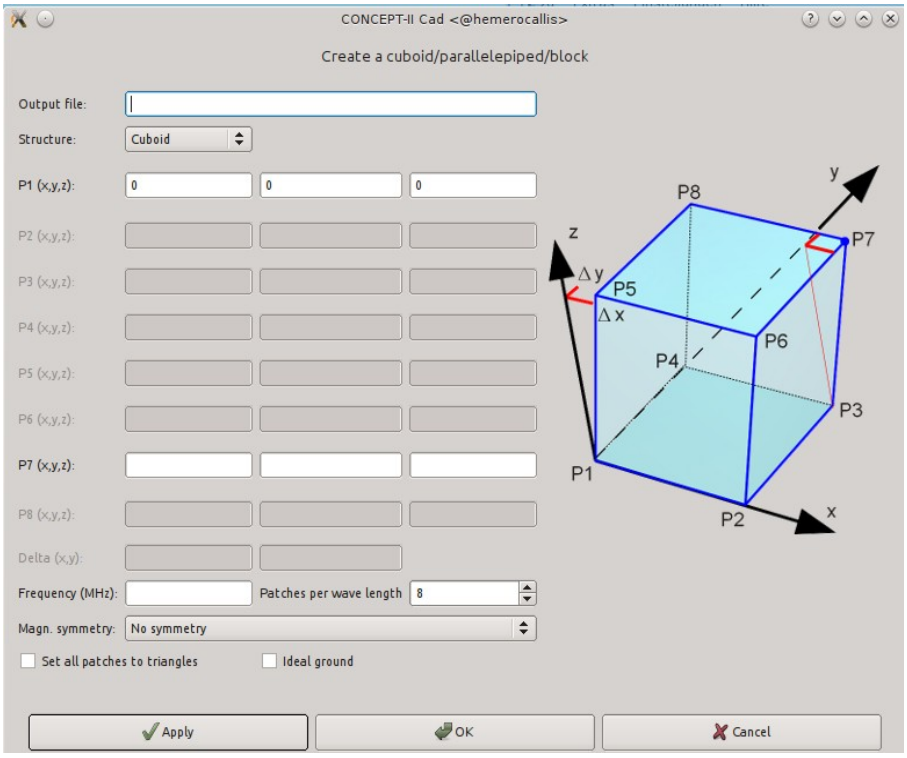


Fig 3.21: In the most general case all eight points P1, …, P8 have to be specified.

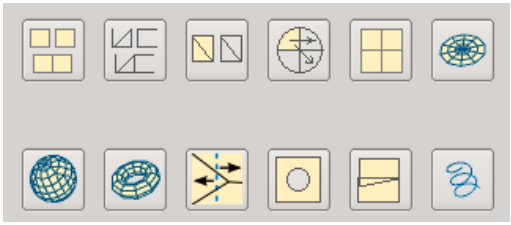
→Meshing of arbitrary circular cylinders., cones, pyramids →Discretization of spheres or ellipsoids → Cubes, rectangular boxes, parallelepiped, plates, pyramids. In case of a cube or rectangular box only P1 and P7 are necessary, for a plate set the z coordinate of P7 to zero.

Fig. 3.21 shows the window for creating arbitrary blocks, cubes etc..

In case of a parallelepiped the two points P1 and P7 have to be specified and in

addition two shift values “Delta (x,y)” ($\Delta x, \Delta y$,see Fig. 3.21). For an arbitrary block the x-y-z coordinates of all 8 vertices have to be entered. A number of special configurations are possible: Setting for example P5=P6=P7=P8 gives a pyramidal structure, an example is shown in Fig 3.22. Setting the z coordinate of P7 to zero provides a plate in the y-z plane.

Further important tools can be found in the bottom section of the **Cad tools 2** window:



(plate)
Discretization of arbitrary plates with a uniform or a non-uniform mesh. The only restriction is that there are straight lines connecting the vertices.

(sphere3, sphere4)
Discretization of spheres, ellipses, etc..

(combi)
Turns a number of individual surfaces into a single surface. Includes also the possibility to repair certain grid errors. The file resulting from the combination process will be shown in the display area.

Surface patch files contain the mesh data, each file with a unique node and patch numbering, starting with node number 1 and patch number 1 (see Section 3.10 for the file format) The goal is now to combine the grids of different surfaces into a single data file, again with a unique node and patch numbering. At a common interface double nodes are sorted out.

(combiw)
Combine different wire structures to a single wire structure

(disk4)
Discretization of

- circular and elliptical disks.
- Disk with a hole in the center
- Cylinders (with different radii at bottom and top)
- cones (also tilted cones)
-

(symgen)
Complete discretized structures. In case of electric and magnetic

symmetry only the symmetric part of the of the overall structure has to be discretized. Applying this tool the complete structure can easily be generated.

Tool to detecting grid errors; tiny patches and very small patches can be found etc. Mesh refinement

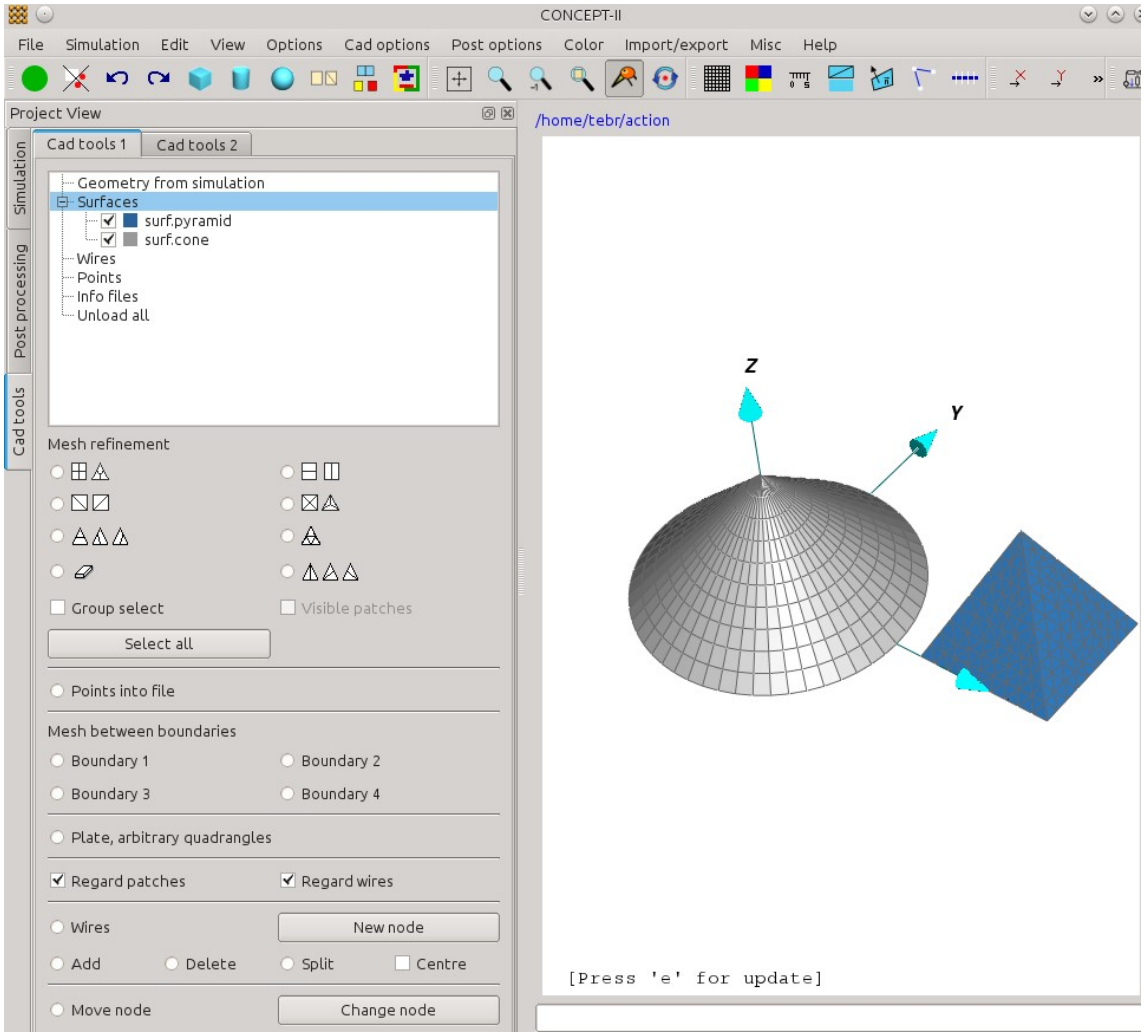
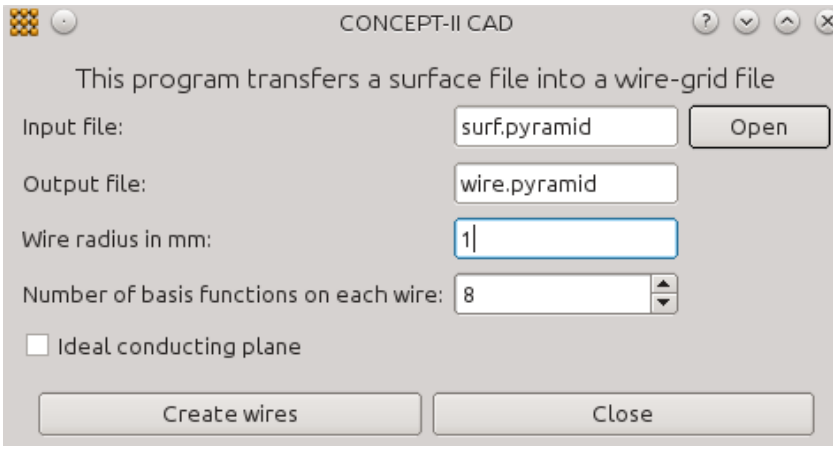


Fig 3.22: **Cad tools 1**: Tools for quick modification if grids, creation of wires and surface parts. The cone has been created by the disk tool and the pyramid with block tool

Surface patch data is converted into a wire grid. All edges are turned into connected wires. The conversion result is displayed in the *Display area* and the

corresponding files name is added to the list of loaded wire files (see the **Cad tools 1** tab). According to the following figure the *Number of basis functions on each wire* and the *Wire radius in mm,* each valid for all wires have to be specified.



Tool for creating rings, helixes, spirals of wires. Currently the output is written into the file *wire.1*.

Tool for reducing symmetrical surface patch structures to the part whichis in the symmetry plane only.

Section **Cad tools 1** (see Fig 3.22)

Activate a preview pattern, for example → move the mouse pointer over a triangle of the surface being displayed in the *Display area* → a right click projects the preview pattern onto the corresponding patch. Typing <e> or clicking on updates the grid.

In case of more than one possibility for patch refinement right click on the same patch again.

Application example 1

The quadrangles marked by black dots in Fig. Fig 3.23a) have shapes which are disadvantageous in some cases already from a numerical point of view. In a good mesh the edges at vertices of triangles should include an angle of approximately 60 degrees and the edges at vertices of quadrangles should have angles of more or less close to 90 degrees. We choose the quadrangle-to-triangle preview pattern and mark the indicated quadrangles. Typing <e> generates the new mesh.

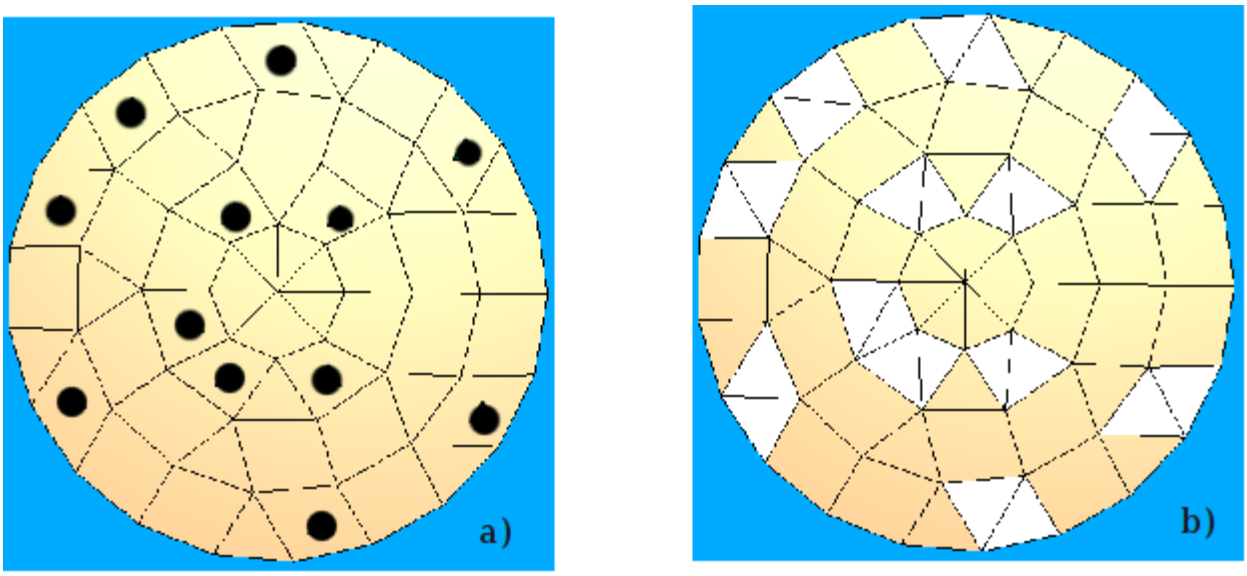


Fig 3.23: Improvement of an existing mesh. a) Original mesh with unfavorable quadrangles. b) Preview pattern

Sometimes it is necessary to change or modify the mesh in large areas. To do this, choose the appropriate preview pattern and activate ☑ Group select, then select the region to be modified: press and hold down the left mouse button and drag the mouse. All patches in the selected rectangular region are marked by the chosen preview pattern.

If ☑ Visible patches is active only those patches are marked by a preview pattern which can be seen on the screen. If not active all patches are modified, even if they are lying behind the front patches.

Click on to rebuild the mesh.

Application example 2

A cubical box according to Fig. 3.24 b) shall be discretized. An aperture has to be inserted into the top plate and the mesh has to be refined around the boundary of the aperture. The following steps are necessary for this task:

1. Create the cube, click on → , edge length 1m. Set the patch size according to the block shown in Fig. 3.24 a) or similar.
2. Select: which halves quadrangles. As two symbols are depicted the pattern can be mapped onto a quadrangle in both orientations.

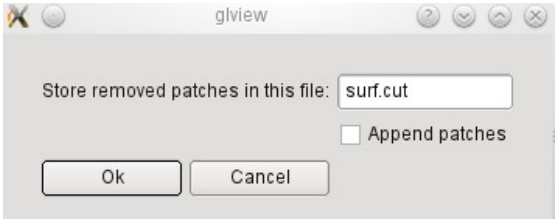
3. Select the top patches according to Fig 3.24 c).

   General rule: If there are several possibilities to place a preview pattern on a patch and the first choice does not have the desired orientation, click again: the next choice will be mapped onto the patch. This feature is also valid in the group selection mode. Once all possible preview patterns were shown the patch (the patches in group selection mode) will be deselected again.

4. Now we have to pay attention to the corners: place the pattern according to as indicated in Fig. 3.24 d).

5. Click on , or type <e>→ The mesh according to Fig. 3.24 e) is generated.

6. Select the rubber tool and click on the center patches of the top plate, see Fig. 3.24 f). Click on , or type <e>→ the following window opens.



   By default surface patches or surface parts that are selected for deletion will be written into the surface patch file *surf.cut.*

   If further patches had to be deleted they would also be written into surf.cut.

   File names for storing deleted Patches can be chosen as follows (*Menu bar*): **Cad options → Store removed patches...**

   It is possible to reload the corresponding surfaces parts: **Cad tools 1 → Surfaces → Load surface file(s)**. It is not possible to take a file name of the list of loaded files for storing deleted patches again. Solution for this problem: **Cad tools 1 → Surfaces →Rename**

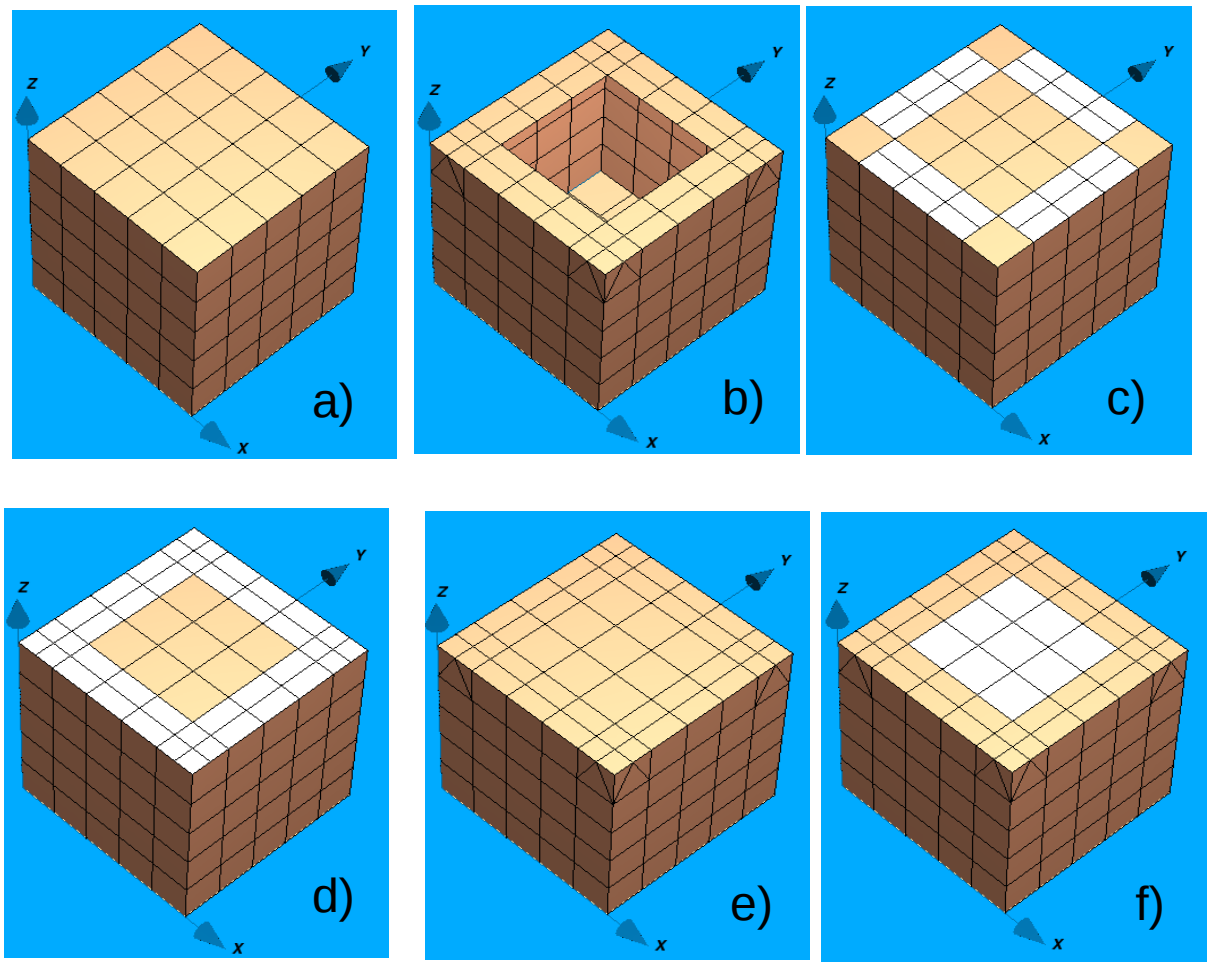


Fig 3.24: How to create a cube with an aperture on the top plate

The box according to Fig. 3.24 b) is generated

Application example 3

Let us assume that the erased aperture part (see previous chapter) has been stored under *surf.2* and that the box has been stored under *surf.1* (see previous chapter and Fig 3.25).

Load surf.1 and surf.2: **Cad tools 1** → **Surfaces** → **Load surface file(s)**

Different colors of surfaces:

- (Menu bar) → **Color** → **Single structure** →select surface file → select color or
- Click on

The mesh of *surf.2* (see Fig. 3.25 a)) shall be refined as it is assumed to be insufficient to cover the coupling into the box improperly.

1. Activate this preview pattern:
2. Select all patches of the aperture as has been done in Fig. 3.25 b) for several patches already.
3. When all patches of the aperture file have been selected for refinement click on → the mesh as illustrated in Fig. 3.25 c) is created. It is important to

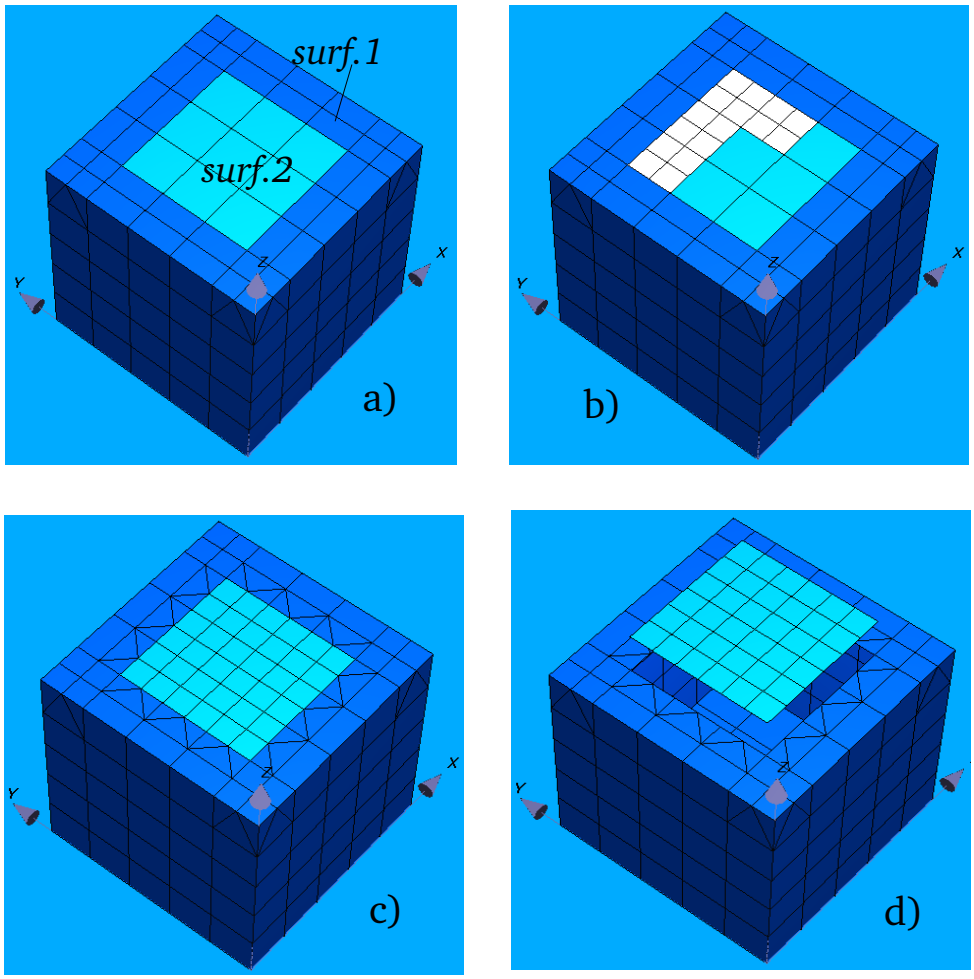


Fig 3.25: Mesh refinement on multiple surface with common boundary

see that although only patches of surf.2 have been selected for refinement the patches at the rim of surf.1 are also effected in order to keep the grid closed. Fig. 3.25 d) shows *surf.1* and *surf.2* when separated from each other. This has been accomplished by: **Cad tools 2** → activate **Move** → enter shift values (*x shift, y shift, z shift*) → select *surf.2* in the file list → type <e> or click on → with (0,0,0.2) we will have the situation according to Fig 3.25

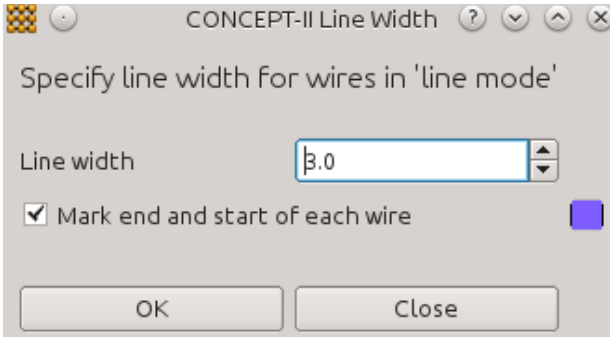
d).

3.6.2 Treatment of wires

Wires can be displayed in two forms

- As straight lines

  This is the default mode when loading wires. Change the line width by



- **Options → Wire line width...** → the following window appears. When “Mark end and start of each wire” is active small dots are placed at the corresponding locations (color blue is chosen here). The representation of dots can be switched on by clicking on .

  Clicking on indicates the wire segmentation according to the chosen

  number of basis functions. Both modes are only available for the line-type representation of wires.

- In 3D cylinder mode. Switch on by: **Options → Draw wires as ... → Cylinders**.

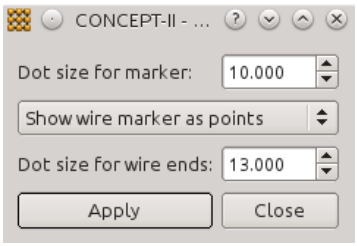
  Attention: in case of small radii the structure might become almost invisible. The 3D character appears when zooming in.

Under the **Cad tools 1** tab the following section exists:

Wires

Once this button is activated wires can be added, deleted or splitted into sub-wires. If the 3D wire representation mode is active it will automatically be switched to the lines mode. The dot size can be adjusted as follows:

Cad options → **Dot size...** → the following window opens:



Add

Wires can be added: select at least two different nodes of the loaded structure by right mouse click and then . The user will be prompted to enter *the Name of wire output file*, *No. of wires between the dots* which have been marked, *Wire radius in mm*, *Number of basis functions on each wire*.

If a new node shall be introduced click on **New node** → a window opens where new nodes can be entered in the top section (see Fig 3.26). Nodes that have been selected already appear under the *List of last selected coordinates*.

Clicking on **OK** adds the entered node with *New coordinates x,y,z* to the *List of last selected coordinates.* Clicking into the fields *X value, Y value, Z value* puts the corresponding coordinates into the top line which can easily be modified.

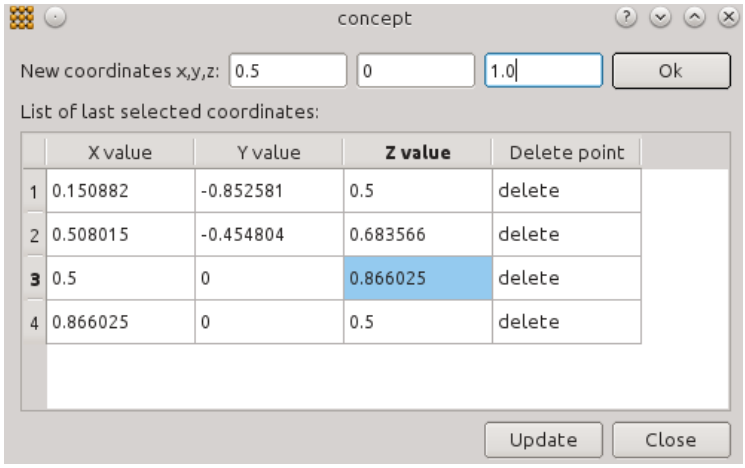


Fig 3.26: Setting of new nodes. Clicking on an existing coordinate value, for example on the high-lighted z value of node 3, causes this value appear in the list of *New coordinates x,y,z* where it can be edited. Editing is also possible by double clicking on the input fields. At least two points must have been specified before clicking on **Update**

At least two nodes are required in the *List of last selected coordinates*. Click on **Update** → a sub-window opens → name of wire file, number of wire segments, radius and number of basis functions have to be specified → the new wire(s) are displayed. In case of a new file name the name is added to the list of loaded wire files.

Delete

Select two or more successive wire nodes and then on → the wire(s) will be deleted. Note that if an unselected node is in between two selected nodes nothing will happen. Click on to check the situation. The default name of the file containing the deleted wires is *wire.cut*. The name can be changed: **Cad options → Store removed wires...**

Split

Right-click on an arbitrary location along a wire → the position is marked. Click on → the wire is split into two subsections. If *Centre* is active, the exact center of the wire is selected, no matter where the user has clicked onto the wire.

Sometimes it is difficult to pick the right node, for example if many wires are in front of a rather fine surface patch grid. Then it can be helpful to switch on the following option:

Now surface patch nodes cannot be selected any more.

3.6.3 Generating triangular and quadrangular plates

When the radio button

is active (**Cad tools 1** tab) arbitrary plates can easily be discretized. The procedure shall be demonstrated by Fig. 3.27 . The top region of the structure according to Fig. 3.27 a) shall be closed, i.e., the final result should look like the one as depicted in Fig. 3.27 f).

- Select the corner points of the top rectangular region as depicted in Fig. 3.27 b) in the indicated order. In the common case marked nodes do not need to assign a flat rectangular region. If only 3 corner points are selected a triangular surface or -in the simplest case- a single triangle is generated, depending on the number of meshes to be specified.

- Click on → a window according to Fig 3.28 opens:

  Enter the output file (*surf.top* in our case). Enter the number of *Meshes* between the nodes, starting with the section between the nodes one and two, then continuing with the number of *Meshes* between the nodes two and three etc..

  Another option would be to enter 'Frequency in MHz' and the 'No. of Basis functions'.

- Clicking on **OK** provides the mesh as depicted in Fig. 3.27 c). *surf.top* has been created and given another color. **Color** → **Single structure** → select *surf.top* → select the color.

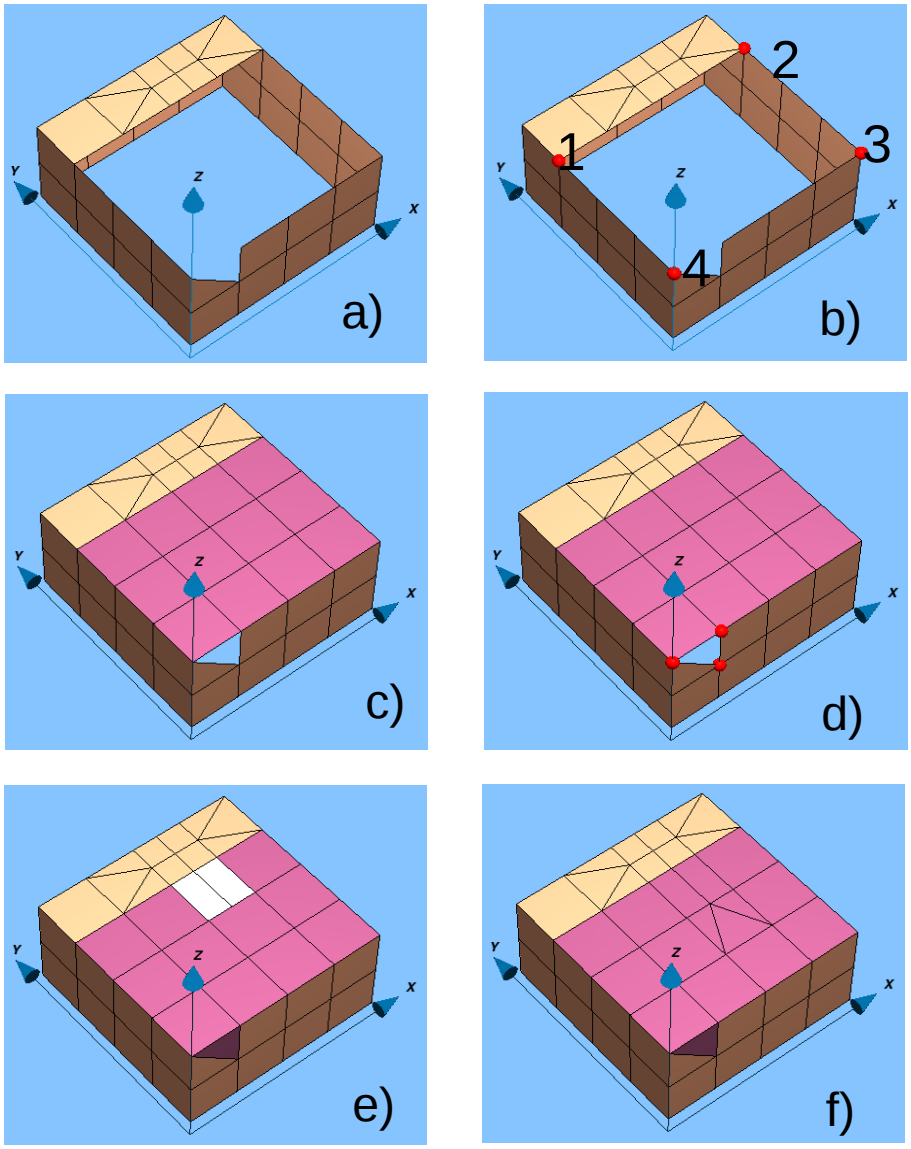


Fig 3.27: Using the plate tool of the **Cad tools 1** section

- As can be seen in Fig 3.27 c) there is still a small triangular region which needs to be closed, hence select the three nodes as indicated in Fig. 3.27 d).

  Under 'Meshes' according to Fig. 3.28 we have to enter four times a “one”, the output file remains the same. Click on → We see the mesh as

  illustrated in Fig. 3.27e).

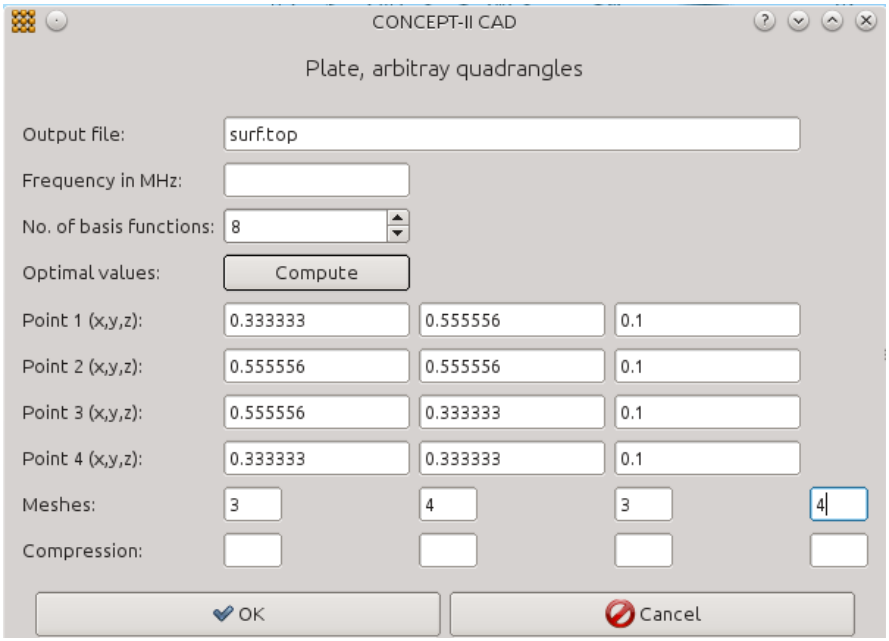


Fig 3.28: Discretizing plates under CAD tools 1 with the plate tool

- The patch grid is not closed. To overcome this grid error we have to refine the grid locally as has been indicated by the preview pattern in Fig. 3.27 e). Clicking on provides the final result.

To create a new plate the following possibilities are available:

1. The nodes of an existing surface patch grid and/or the nodes of wires can be selected. There is no need that such wires belong to the structure under investigation, they may simply serve for marking the outline of a structure part or likewise. Only wires which have been imported into the **Simulation** section are part of the structure to be analyzed by the MoM.
2. Points in space can be created and selected. For such a purpose activate (**Cad tools 1** → **Points** → **Edit)** A window (similar) to the one shown in Fig.3.29 opens.

   Here the coordinates of points in space can be specified. Such points are shown as small spheres in the display area. Points can be taken from the structure by right-clicking on nodes.

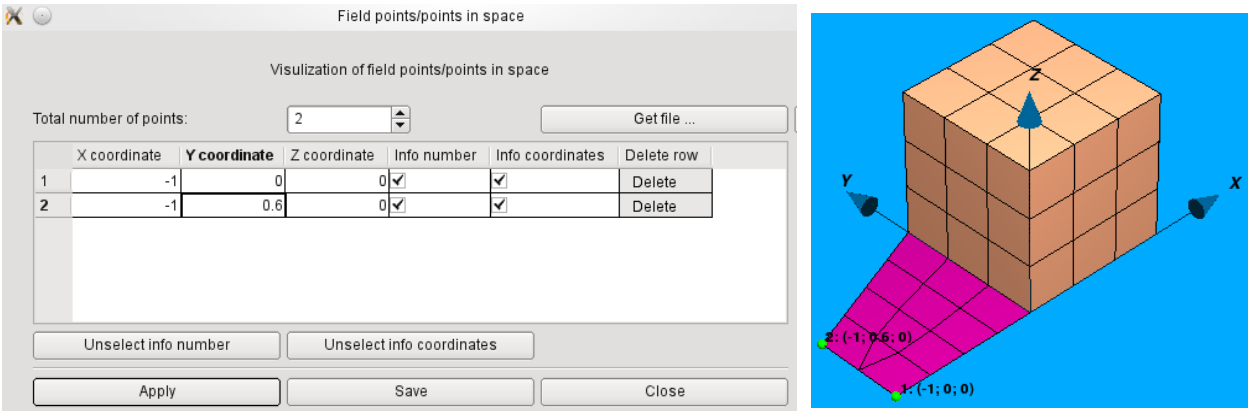


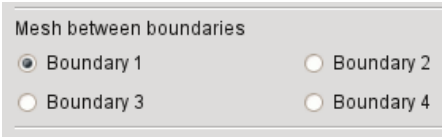
Fig 3.29: Left: A window for entering arbitrary points in space with 2 example points. Right: These points were used to discretize the additional bottom plate.

Note: A plate tool with an enhanced number of features is available under **Cad tools 2** , .

3.6.4 Using the boundary tool

Surfaces with curved boundaries can be discretized with the boundary tool:

Fig 3.30: The boundary tool under the **Cad tools 1** tab)



The required steps for discretizing a surface with two irregular boundaries is explained by Fig. 3.32. The missing surface part within the half disk has to be filled by an appropriate pattern of surface patches. (Fig 3.32 a))

- Activate the radio button “Boundary 1” (see Fig. 3.30) and select the nodes of the first boundary. Pay attention that the nodes are selected in a row. We get the sequence of blue marked nodes in Fig. 3.31b).

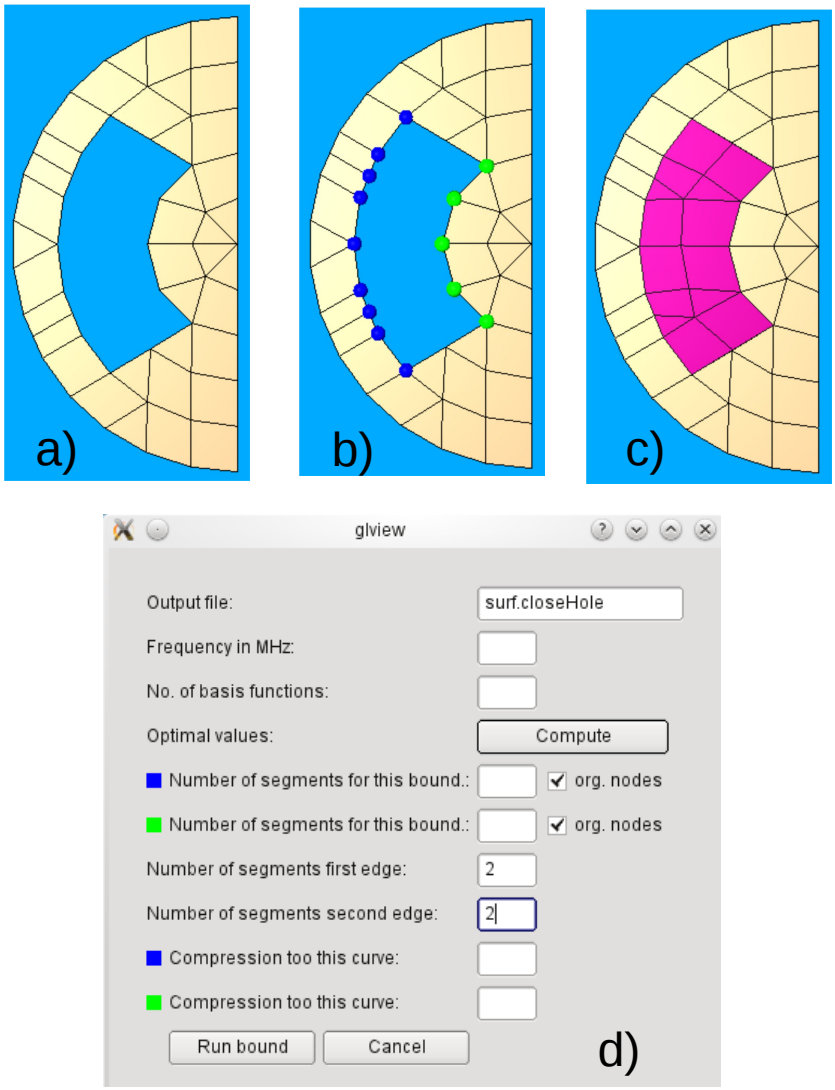


Fig 3.31: Working with the boundary tool

- Activate the radio button “Boundary 2” and select the nodes of the opposite boundary - in our example we have the green marked nodes. It does not matter which boundary is selected first and it is not important at which corner the selection is started.

- Click on ● or type <e> → the window according to Fig. Fig 3.32 d) opens.

  The grid shall be perfectly closed, therefore we activate the two check boxes **org. nodes** and set two segments at each of the remaining boundary sections.

- Clicking on **Run bound** closes the aperture by a surface patch grid as illustrated in Fig. Fehler: Referenz nicht gefundenc)

Hint: Using the boundary tool the coordinates of selected nodes are immediately

written into ASCII files. Boundary 1 is active → the file *bound.1* is generated, Boundary 2 is active → the file *bound.2* is generated. This may be helpful for many purposes, for example for a copy and paste action of complicated coordinates. These mentioned files can easily loaded by **Cad tools 1 → Points → Load point file(s)**

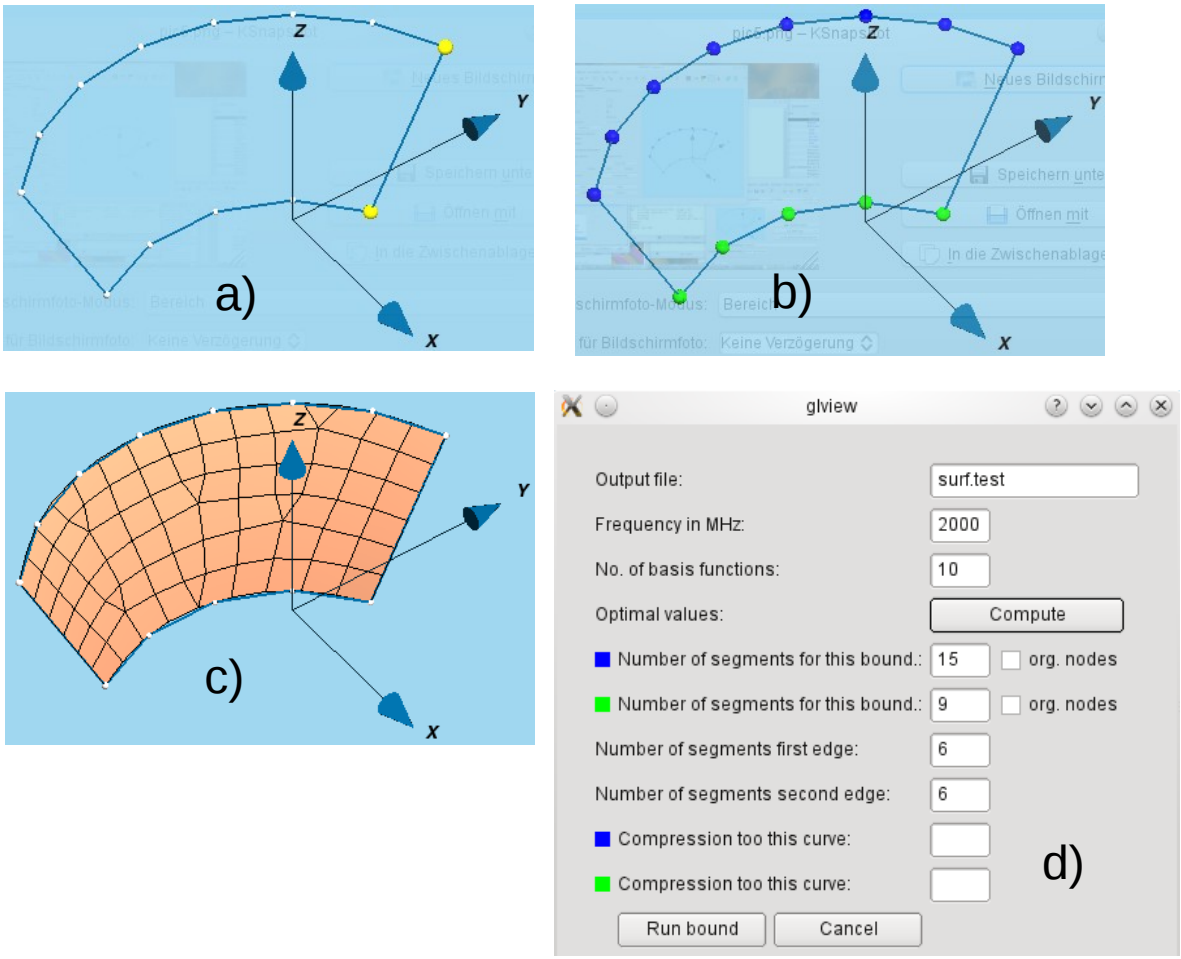


Fig 3.32: Working with the spline mode of the boundary tool

Another option for using the boundary tool shall be demonstrated by means of Fig. 3.32. The task is to discretize the area which is bounded by straight sections (wires) as illustrated in Fig 3.32a). The assumed frequency is 2 GHz and the distance between the yellow dots is 8 cm. Therefore the intervals between the white dots designating the end of the sections are too large. To get a grid which obeys the 10-patches-per-wavelength rule of thumb we can proceed as follows:

- Mark the upper and lower curved boundaries (see Fig. 3.32 b)) as has been described in the previous example

- Click on '→ the window according to Fig. 3.32 d) opens. Now enter the

  Frequency in MHz and the *No. of basis functions* (per wavelength) and click on **Compute** → the number of segments along all 4 boundaries are evaluated automatically.

- Click on **Run bound** → the mesh according to Fig. 3.32 c) is generated. Notice that the original nodes are not used any more; i.e., the curved boundaries are approximated by spline functions and the necessary nodes have been equidistantly inserted.

- Be careful when generating grids in the same way on adjacent surface parts which might be galvanically connected to the depicted curved surface. It is strongly recommended in such a case to check grid by means of the grid check tool to find out if the grid is closed. If small gaps appear, the grid can easily closed by means of the combine tool (see **Cad tools 2**). Set *Distance between nodes* in an appropriate manner.

3.6.5 Effective discretization of wire contour models

In practical cases the contour data of a physical structure may be available from CAD systems and turned into the *CONCEPT-II* wire format. An example is depicted in Fig. 3.33 a). We have the outline of a ship; due to symmetry only half the model needs to be considered setting up the surface patch model (see *$CONCEPT/examples/ship/shipsym/ship.wire*).

The following steps are suggested to generate a mesh suitable for a maximum frequency of 30 MHz, part of the final model can be seen in Fig. 3.33 b).

1. Load the wire-contour model and click on .Clean and complete the wire structure: useless points should be deleted. This can be accomplished by activating the **Move node** button under **Cad tools 1**. Mark a node which shall be deleted → right click on the next neighboring dot (wire end) → the corresponding wire disappears.

   Additional points should be inserted at places where they are necessary: use the split-wire option. Use **Cad tools 1** → **Points** → **Edit** to create points in space.
2. Quadrangular or triangular sub-surfaces can be generated as has been described in Sec. 3.6.3.

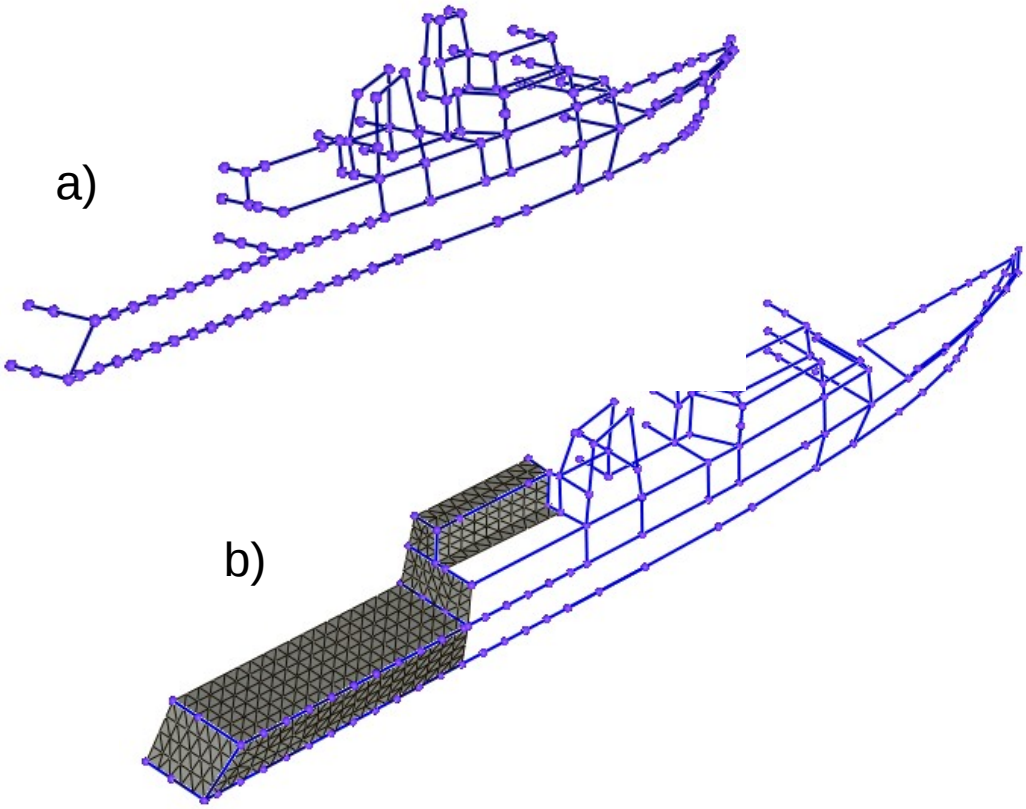
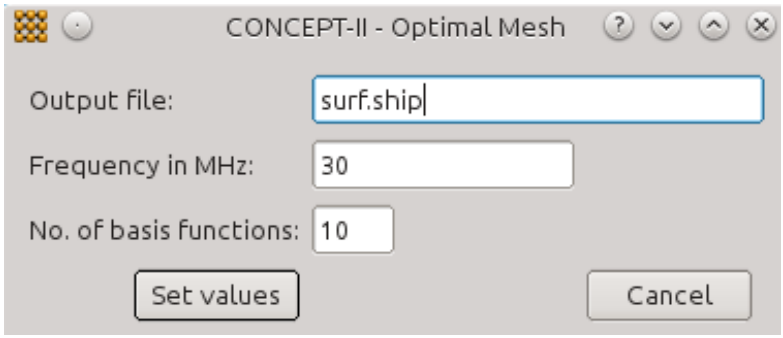


Fig 3.33: Using a wire grid model for generating a surface patch model. The original mesh of the depicted surface parts was mixture of quadrangles and triangles. A mesh of only triangles was generated by clicking on (*Tool bar*). See *$CONCEPT/examples/ship/shipsym).*

3. The discretization of sub-plates can be accelerated by two measures:
   a) **Cad option**s → '**Values optimal mesh...** → the following window opens:



   Notice the entries in the input fields. The overall length of the model under consideration is 72 m.

   b) **Cad options** → **Use optimal mesh**

The effect is that when having selected the points, the plate will immediately be generated once the user has entered <e> (character “e”).

For the rapid positioning of the mouse pointer turn the mouse wheel for quickly zooming in. Press the <CTRL> key + click the left mouse button and drag to shift the depicted structure in the display area. Turn the mouse wheel in reverse direction to zoom out again. Once all sub-plates have been introduced it is recommended to check the grid. Click on

(**Cad tools 2**). A window opens. Activate the check box **Find free nodes**. Click on **Apply** → the contour of free nodes is shown in wire format (file: *wire.freenode*). Line sections are only allowed to appear along open boundaries. When they appear at places where the grid should be closed, there is an error – the corresponding nodes are separated and the *CONCEPT-II* back end will not be able to compute currents crossing that line. Once the mesh has been generated and errors have been repaired, use the following tool (**Cad tools 2**) to complete the structure for the

full ship

Introduce an HF antenna as has been described in Sec. 3.6.2, generate a local mesh refinement at the base of the antenna is necessary

Load the structure into the **Simulation** section ...

3.6.6 Using the point option for discretizing

In the last section the outline of a ship has been used for generating a surface patch grid. Such a task could also be performed by making use of points in space, which shall be demonstrated in another simple example. In Fig. 3.34 a transmission line (wire loop) on a square plate is shown.

For lines over a discretized ground it is important that the typical size of patches underneath the horizontal conductor is at least approximately equal to the height of the conductor.

Although this condition is approximately fulfilled in the surface grid according to Fig. 3.34a) **we see a grid as it never should be. This grid is extremely bad. We cannot expect good computational results from this.** How can we improve this? The following proceeding is suggested:

- Mark important points, vertices etc. as indicated in Fig. 3.34 b): **Cad tools 1 → Points → Edit a new point file.** Enter the name of the point file select nodes by right-click. **Apply** or **OK** saves the file.

  Hint: Clicking 'somewhere' into the region of a patch takes the corresponding coordinates. If a node is in the focus, the coordinates appear in the output line under the *Display area*.

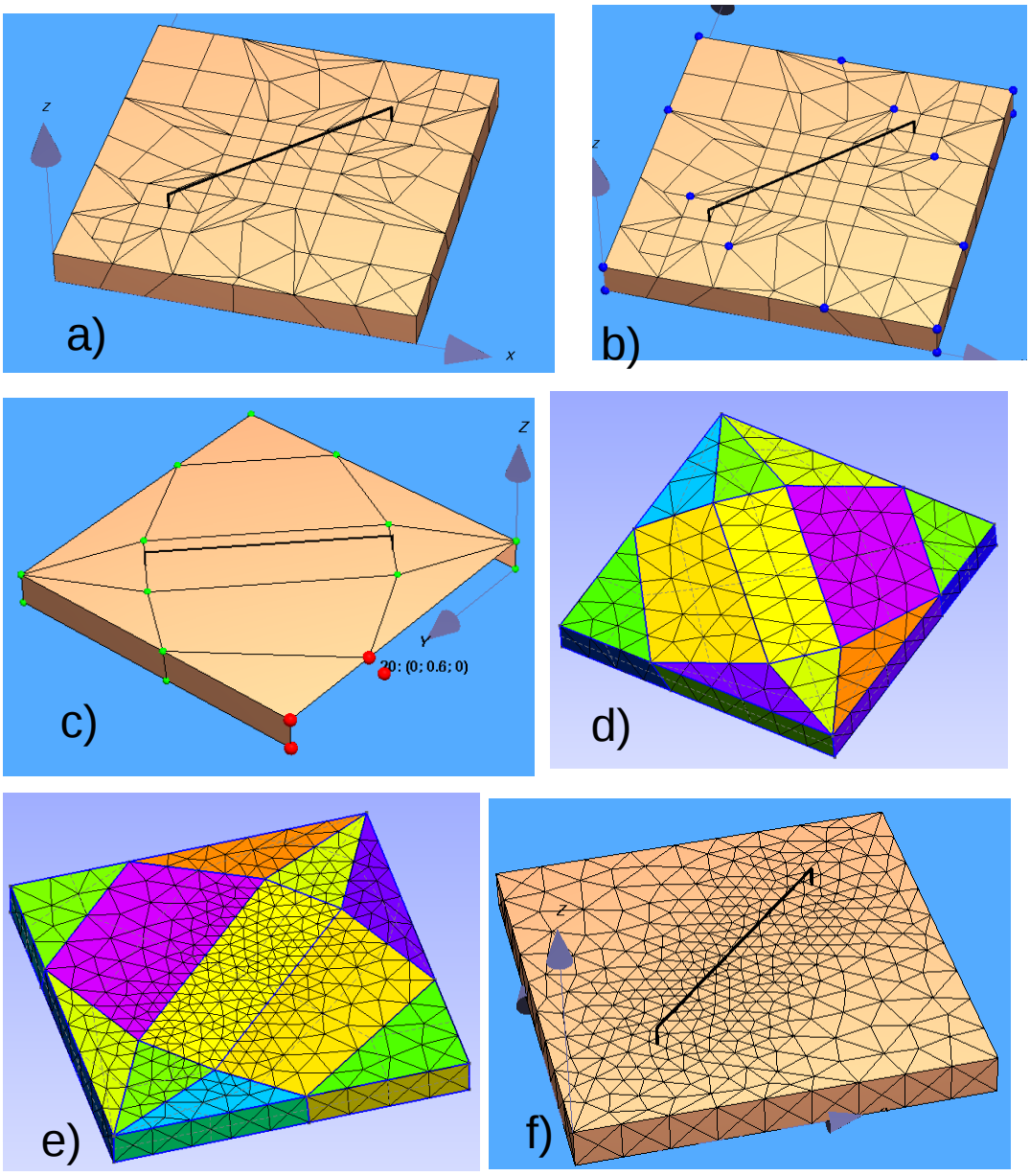


Fig 3.34: Making a good grid out of a bad grid using the *CONCEPT-II* **Import/export** tool in conjunction with *gmsh* (see Sec. 1.6)

- Unload the surface patch file containing the ugly grid; we have the points and the loop on the screen. Based on the pattern of points generate a rough plate grid as can be seen in Fig. 3.34 c). Proceed as described in Sec. 3.6.3. assuming only a single mesh element on each side. Save this file.

  This data shall now the used to create a nice, frequency-dependent grid.

  Gmsh shall be used to for this purpose (see Sec. 3.7 and Sec. 1.6).

  2Menu bar → **Import/export** → the Import/export dialogue window opens → proceed as described under Sec. 3.7.3. The resulting surface patch grid is displayed in the *gmsh* window as illustrated in Fig. 3.34d). The assumed frequency was 300 MHz, the square plate is 1m x 1m and the *characteristic edge length* according to *gmsh* has been set to 0.1m. The height of the horizontal wire is 5 cm, hence an additional grid refinement is necessary along the course of the line.

- Start *gmsh* outside the CONCET-II GUI. *Gmsh* → "Mesh' → 'Define' → 'Element size at points' → window opens →set element size to 0.04 → select corner points of yellow center section by left mouse clicks and close this action be typing 'e'

  'Geometry' → 'Reload' → 'Mesh' → '2D' → the grid according to Fig. 3.34 e) appears.

  Before remeshing the grid it could be advantageous to set: 'Tools' →'Options' → 'Mesh' → '2D algorithm' → 'Delaunay'

  Save the data in .stl or .msh format.

- Turn back into the *CONCEPT-II* GUI and convert the .stl or .msh file into the *CONCEPT-II* surface patch format using the **Import/export** tool. We get the structure as can be seen in Fig. 3.34 f). Notice the improvement when comparing with Fig. 3.34 a)!

- Practically we might have the situation that the vertical wire does not hit a patch node, only handle wire-node junctions can be handled. This can very easily be corrected:

  - Move the nearest patch node to the wire node (**Cad tools 1** → **Move node** → select node where to move)
  - Introduce an appropriate grid refinement at the junction point as is demonstrated in Fig 3.35a), see also the application examples in Sec. 3.6.1 .

- An example of the current distribution is shown in Fig. 3.35b) , a generator is at the left vertical wire and load at the opposite vertical wire.

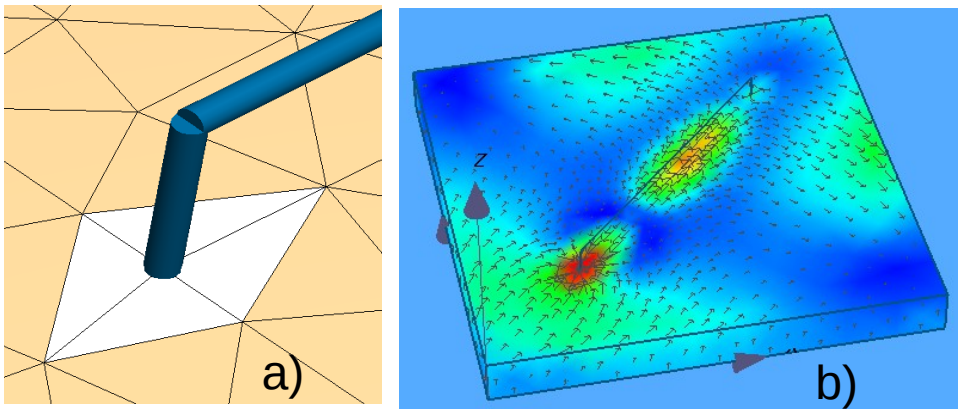


Fig 3.35: Mesh adaption at vertical wires and current distribution

3.6.7 Joining surface parts with unwanted gaps

It might happen that the nodes forming the boundary of different adjacent surfaces which shall be combined to form a single surface do not match (combine tool ,

Cad tools 2). Let us suppose that we have separately discretized half of a cylinder and half of a disk and combined these parts to form the structure as depicted in Fig. 3.36 a). We immediately recognize the grid error at the rim of the disk section. In the general case of a complicated grid the situation might not be so obvious and we can apply the grid check tool (check box **Find free nodes** active).

In our simple example the check tool provides the file *wire.freenodes* which is automatically loaded into the **Cad tools** section, see Fig. 3.36 b); dots are marking the wire ends.

The following proceeding is suggested to repair the mesh:

- **Cad tools 1** →

→ Right click on a node to be shifted, right click onto the position where the selected node shall be shifted to.

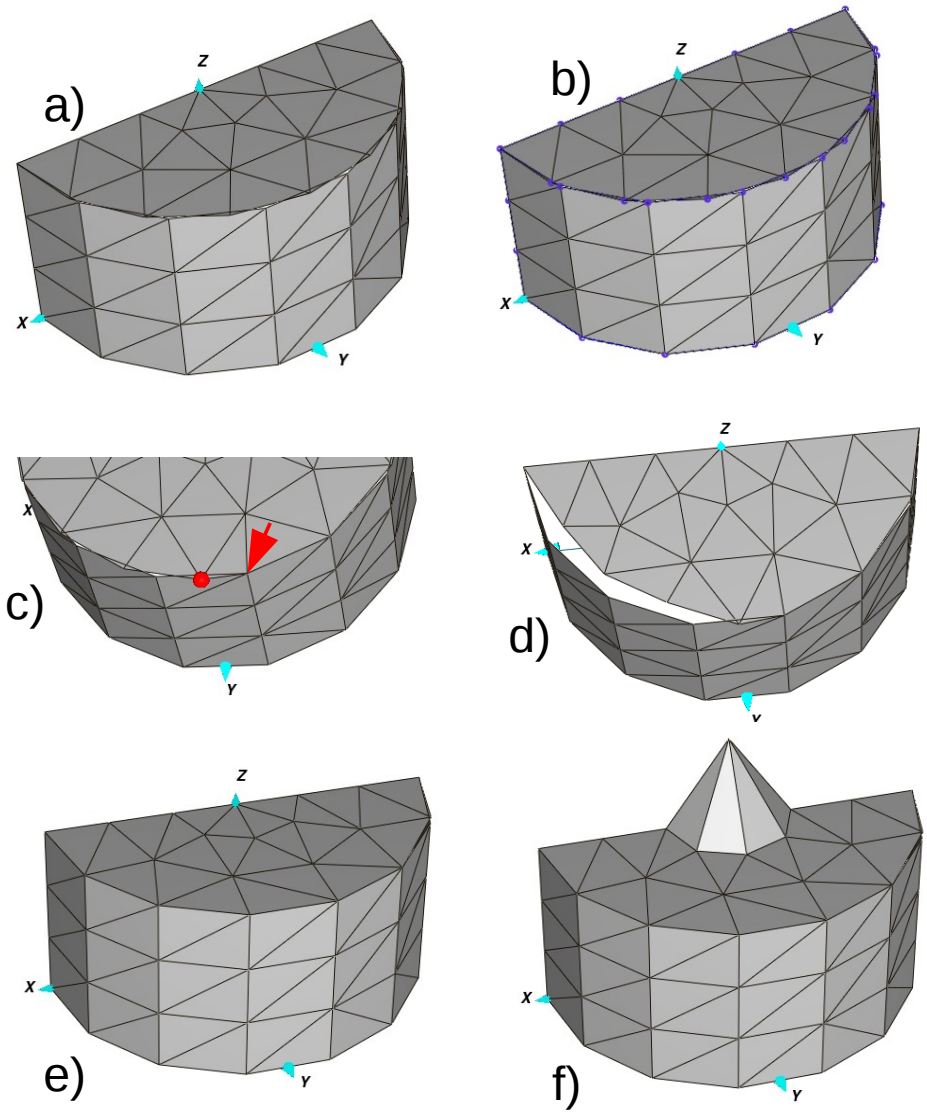


Fig 3.36: Surface patch nodes do not match at the interface line between the top half disk and the half cylinder.

a) Grid is not closed along the rim of the top disk. b) Result of the grid check tool. c) Shifting the red dot to the position the arrow is pointing to will erase the top triangle automatically. e) the gap has been closed. d) shifting a node by the **Change node** option.

In some cases it is advantageous to make the gap even larger by first shifting the nodes clearly “to one side” as illustrated in Fig. 3.36 d) before closing the gap. In Fig 3.36c the red dot marks the node which shall be shifted to the position where the the red arrow is pointing to. By this action the corresponding triangle will automatically be deleted.

Hint: If unsure if a single node exists or 2 (or more) nodes hiding each other, shift the doubtful node by a small distance into one of the adjacent patches. If

no gap opens there is only a single node. Click on to get the initial situation back again.

- The coordinates of nodes can be changed to positions which are not clickable. Select a node (example: Fig. 3.36 e), coordinates: (0,0,0.1). Click on **Change node** → a window opens, enter the new node coordinates →**OK**→ the node coordinates are immediately changed, in our example to (0.,0.,0.15), see Fig. 3.36f).

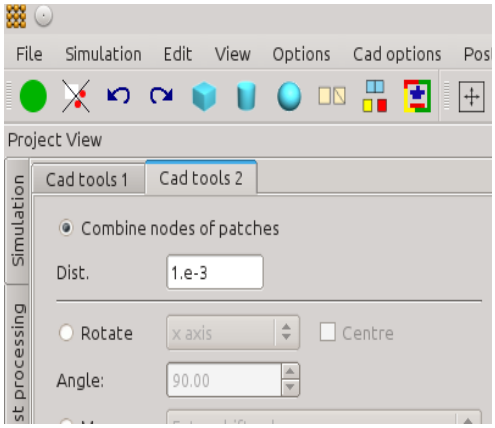


Fig 3.37: A convenient possibility to achieve a closed grid in case of small gaps

Imagine the case of several surfaces forming a body. Let us assume that the grid is not closed as the nodes along the boundaries of adjacent surfaces are not lying on top of each other as has been detected by the grid check tool; we have a situation similar to the one depicted in Fig. 1.6 with the gap hardly being visible by inspection. Under such circumstances we can use the option according to Fig. 3.37, *Combine nodes of patches* is active. All nodes will be combined to a single node once the distance between them is small than 1 mm in the example. All loaded surface patch files (see Cad tools 1) will be investigated.

3.7 Importing and exporting

The STL format and the *gmsh*-specific .msh format can be converted into the *CONCEPT-II* surface patch format (for *gmsh* see Sec. 1.6 and below).

In case of other CAD formats: *gmsh* can import IGES and STEP data and can convert these data into the .msh or the STL format.

3.7.1 Converting STL data into surface patch format

This can be accomplished by clicking **Import/Export** → A window as illustrated in

Fig. 3.38 opens. The STL format is supported by many CAD software packages. The STL format refers to triangulated surfaces of three dimensional objects. The format specifies ASCII and binary representations, both versions are supported. The binary format is more compact.

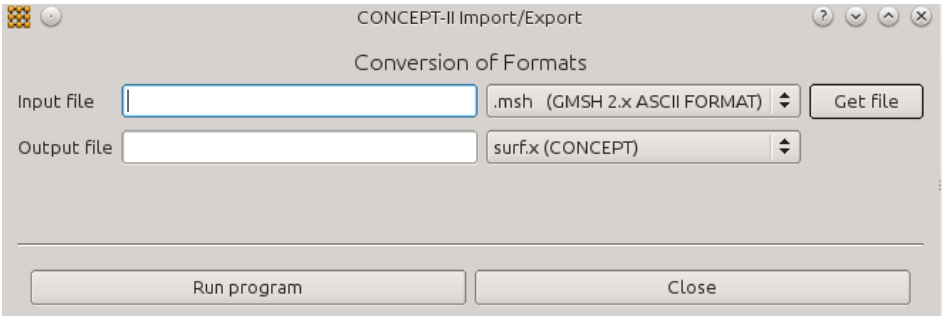


Fig 3.38: Format conversion in *CONCEPT-II*

The procedure of data conversion shall be demonstrated by of examples. The program *gmsh* is available on the Internet under the following address:

(http://geuz.org/gmsh/).

It is distributed under the conditions of the General Public Licence (GPL) [C.Geu-2009] and in no way a part of *CONCEPT-II*.

Suppose that we want to model a triangulated plate with circular aperture as shown in Fig. 3.39 (*gmsh* display area).

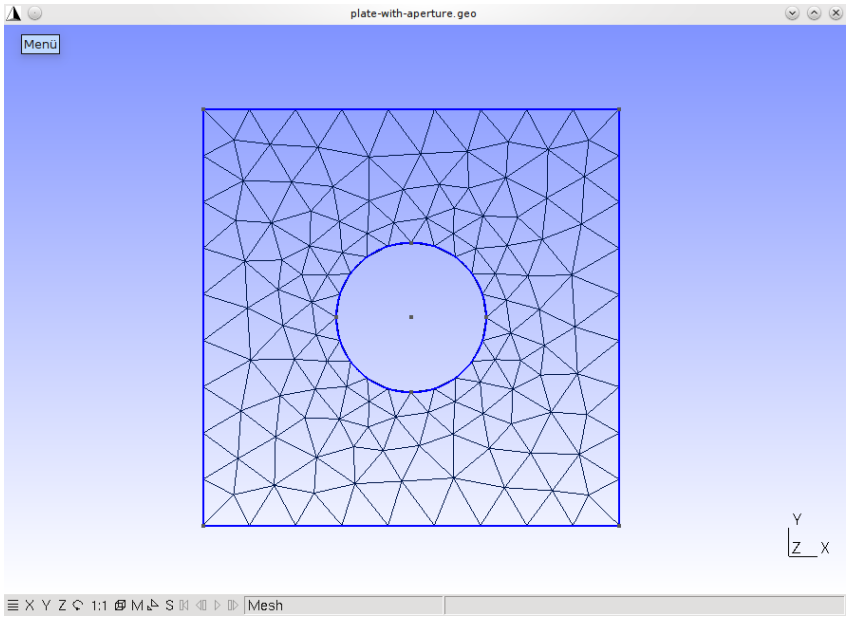


Fig 3.39: A triangulated plate with a circular aperture (loaded file *plate-with-aperture.geo*)

For the application of *gmsh* please study the respective documentation, examples are on the web page, see above. In our case the loaded file *plate-with-aperture.geo* looks like this:

```
// Surface patch model of a plate with hole
// Characteristic length of outer plate boundary
clplate=1;
// width, height of the plate
b = 10;
h = 10;
// Vertices of the plate
Point(1) = {0.0, 0.0, 0, clplate};
Point(2) = {b, 0.0, 0, clplate};
Point(3) = {b, h, 0, clplate};
Point(4) = {0.0, h, 0, clplate};
// Aperture
// Radius of aperture
rap =1.8;
// characteristic length
clap=0.5;
// Center point of aperture
Point(9) = {b/2, h/2, 0, clap};
// 4 points around the center of the circular aperture
Point(10) = {b/2, h/2+rap, 0, clap}; Point(11) = {b/2, h/2-rap, 0,
clap};
Point(12) = {b/2+rap, h/2, 0, clap}; Point(13) = {b/2-rap, h/2, 0,
clap};
Line(1) = {4, 3}; Line(2) = {3, 2};
Line(3) = {2, 1}; Line(4) = {1, 4};
Circle(5) = {12, 9, 10}; Circle(6) = {10, 9, 13};
Circle(7) = {13, 9, 11}; Circle(8) = {11, 9, 12};
Line Loop(9) = {4, 1, 2, 3};
Line Loop(10) = {6, 7, 8, 5};
Plane Surface(11) = {9, 10};
~
```

Open by
> gmsh plate-with-aperture.geo <Return>

Note that it is very convenient to modify the plate size (parameters: b,h) or the radius of the aperture including the mesh density at the rim of the aperture (parameters: rap, clap). *Gmsh* can export its *.msh* format into the STL data format → *plate-with-aperture.stl* is generated. The next figure illustrates the entries to be made (see Fig. 3.38, **Import/export** tool).

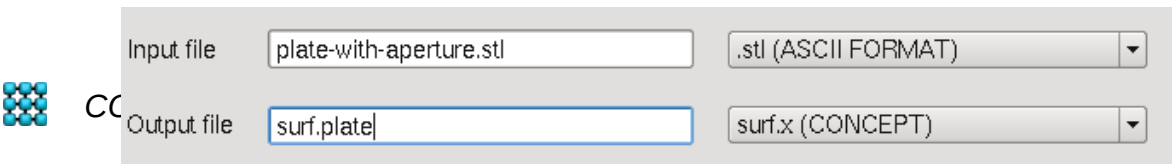


Fig. 3.40 depicts the result of the conversion. The left surface patch mesh is based on the contents of the *plate-with-aperture.geo* file as printed out above. For the right mesh the position of the aperture (Point(9)) and the mesh density at the aperture boundary (clap) have been changed.

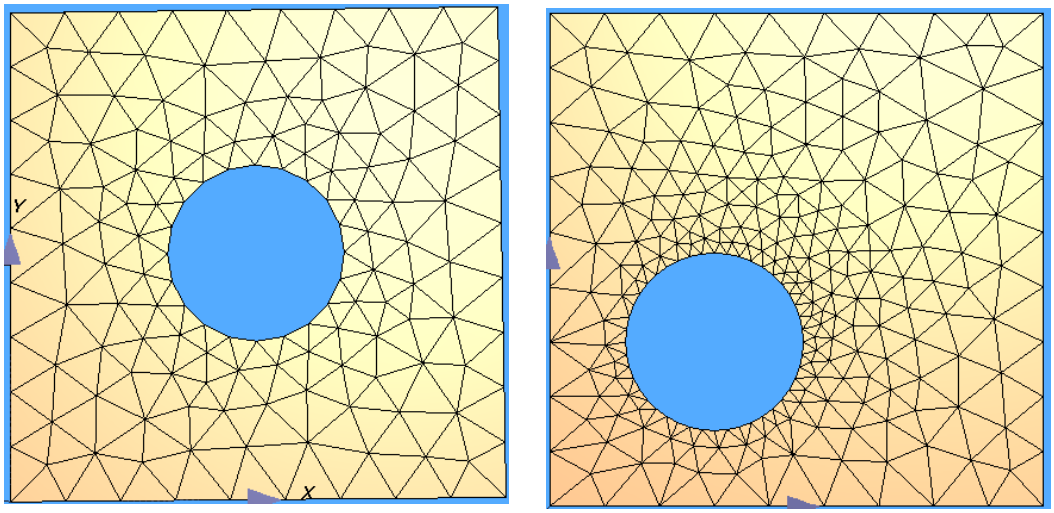


Fig 3.40: ASCII STL files have been converted into the *CONCEPT-II* surface patch format

3.7.2 Converting the wire format into the .geo format of *gmsh*

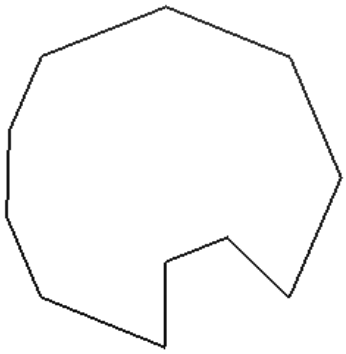


Fig 3.41: Complicated outline of a surface described by wires (*CONCEPT-II* wire format). Discretization of such a plate would be possible by means of the *CONCEPT-II* tools but is inconvenient already.

The procedure shall be demonstrated by means of an example: let us suppose that the outline of a special plate structure looks as illustrated in Fig. 3.41 .The depicted region shall be triangulated.

Once the outline of an arbitrary structure is described by wires (*CONCEPT-II* wire format) the corresponding file can be converted into the .geo format of *gmsh*.

Let the wire file name of the example boundary according to Fig. 3.43 be *wire.outline* . Then the entries as illustrated in Fig.3.42 are required to generate the .geo file.

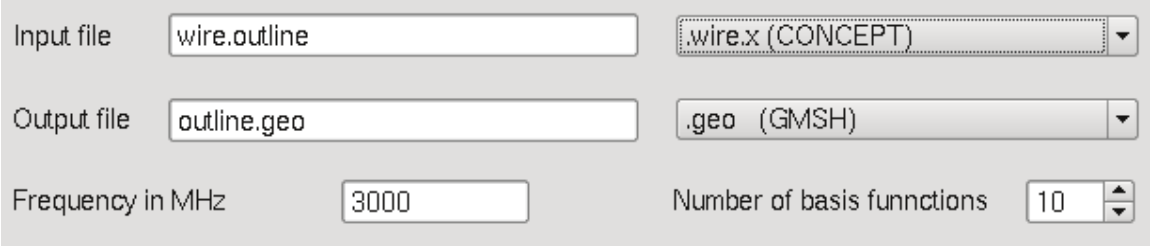


Fig 3.42: Entries for a the conversion of an outline file given in the *CONCEPT-II* wire format to the .geo format of *gmsh*

The diameter of the contour as shown in Fig. 3.41 is 0.2 m, hence setting the frequency to 3 GHz will provide a rather fine mesh already. The “Number of basis functions” can be set to an arbitrary integer value. Running conversion tool provides *outline.geo*, which is an ASCII file, which can easily be modified by an editor, for example setting the frequency to another value.

At the prompt:

> gmsh outline.geo <CR> → the *gmsh* window opens, showing the outline.

Continue with: 'Geometry' → 'Elementary entities' → 'Add' → New →Plane surface →select the boundary by mouse click →type e <CR> → it is indicated that a surface is ready to be discretized (dashed lines) →'Mesh' → '2D'

The mesh as shown in Fig. 3.43 is generated. The grid can be exported in STL format and can then be converted back into the *CONCEPT-II* surface patch format as has been described in the last section.

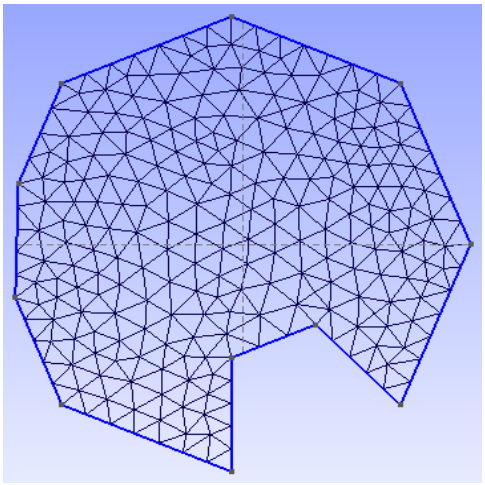


Fig 3.43: The triangulated plate according to Fig.

Note that arbitrary wire files can be converted into the .geo format. Performing such a step, wires have nothing to do with conductors but are used as a means for describing the outline of structures.

3.7.3 Using the *CONCEPT-II* patch format for mesh generation in *gmsh*

Surface patch files of *CONCEPT-II* format can be converted into the .geo format of *gmsh*. Again, let us consider an example. The structure according to Fig. 3.44 can be found under *$CONCEPT/examples/ship*. The length of the ship is around 72 m.

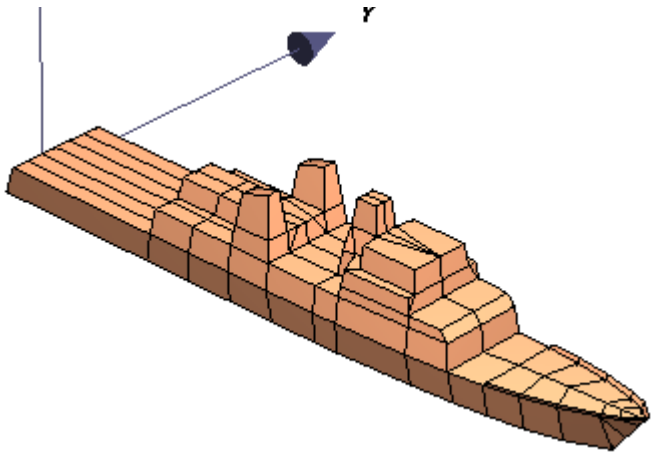


Fig 3.44: A very rough surface patch model of a ship. How to set up such structure has been described in Secs. 3.6.3 and 3.6.5

Let us suppose that we want to have a patch grid which is suitable for an antenna excitation at 100 MHz. A triangulated model is required for this frequency assuming

8 basis functions per wavelength, output file: *ship.geo*, see Fig. 3.45. Click on 'Run program'.

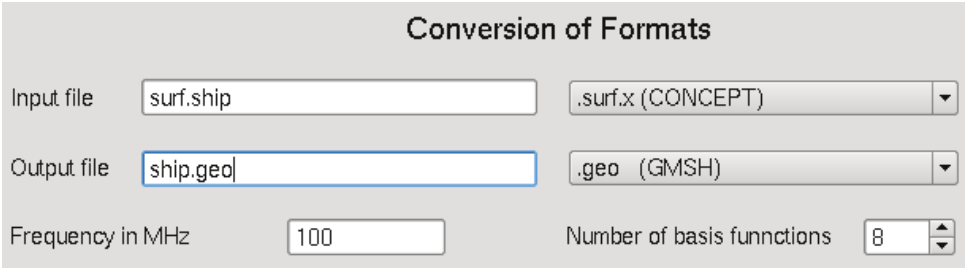


Fig 3.45: *CONCEPT-II*, patch conversion to the .geo format of *gmsh*

Continue with:

gmsh ship.geo <CR> (or start *gmsh* and load *ship.geo*) → 'Geometry' → 'Elementary entities' → 'Add' → 'New' → 'Plane surface' → all surface are indicated by crossing dashed lines → 'Mesh' → '2D'

Export the model as STL file. Convert to a *CONCEPT-II* surface patch file and load into the *Display area.* A part of the triangulated model of the ship is depicted in Fig. 3.46.

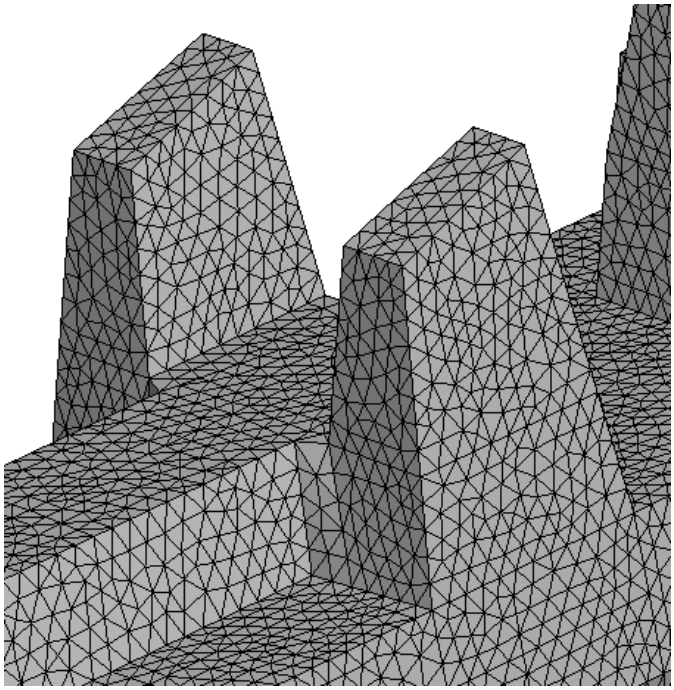


Fig 3.46: Triangular mesh for 100 MHz in the funnel region

At the the prompt:

gmsh -2 ship.geo <CR> → *surf.msh* is generated (*gmsh* surface patch format: .msh).

Convert to the *CONCEPT-II* patch as shown above.

From the outlined example follows that it is easy the generate a mesh refinement for any higher frequency based on an arbitrary existing surface patch model.

The head section of the .geo file under discussion looks like this:

```
//Speed of light
vv00=299.7922517;
//Frequency in MHz
freq=100;
//Number of basis functions per wavelength
nzsw=8;
well=vv00/freq;
len=well/nzsw;
P1=newp;
Point(P1)={18.060499, -4.681880, 0.000000, len};
P2=newp;
Point(P2)={18.060499, 4.681880, 0.000000, len};
P3=newp;
.
.
.
```

As can be seen the characteristic length is set according to the frequency (freq) and the number of basis functions per wavelength (nzsw).

3.7.4 Visualizing field and current distributions by means of *gmsh*

Field and current distributions which have been created in the post-processing stage can be displayed by *gmsh (*version 2.6 or newer, experimental stage, July 13). The data files are only available if the corresponding *CONCEPT-II* tools have been finished:

gmsh co_cmovie.msh <CR> → display current distribution. Tool must have

been finished

item **Post-processing** → entries **[1]**, **[2]**

Tools → **Options** → **View[0]** → *Intervals type* → **Logarithmic …** → Color scaling

View[1] → *Intervals type* → L**ogarithmic …** → Vector scaling

Many options are available for professional looking graphics files. For details refer to the *gmsh* documentation.

gmsh eh2d.msh <CR> → 2D field computation must have been finished

gmsh co_fmovie.msh <CR> → Field distribution must have been displayed for a single phase or several phases.

[C. Geu-2009] C. Geuzaine and J.-F. Remacle. Gmsh: a three-dimensional finite element mesh generator with built-in pre- and post-processing facilities. International Journal for Numerical Methods in Engineering, Volume 79, Issue 11, pages 1309-1331, 2009

3.8 Time domain system responses by an IFT

CONCEPT-II is based on frequency-domain integral equations (see Sec. 4). For time-domain quantities it is necessary to compute a series of sample frequencies in a frequency loop and to perform a subsequent inverse Fourier transform (IFT). Choosing the basic frequency, the highest frequency and the sampling rate in the right way is essential for a successful simulation. Hence some preliminary considerations are recommended as described in detail in Sec. 3.8.2.

3.8.1 The necessary steps in short

1. Discretize the structure in the usual way. Power generators are not allowed.The key point for a successful application of an IFT is the frequency sampling **Simulation** tab → **Frequencies** → **Frequency interval for**

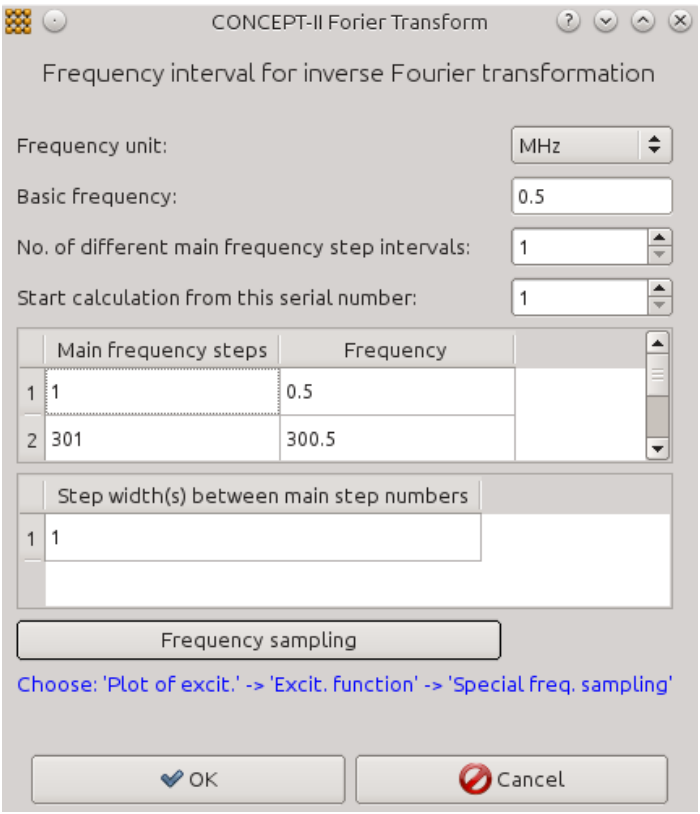


Fig 3.47: Frequency sampling for an IFT

inverse Fourier transform → a window according to Fig 3.47 opens.

2. At the beginning set the following items to 1: *No of different main frequency step intervals*, *Start calculation from this serial number*, *Step width(s) between main step numbers*
3. All remaining input data have to be completed in a the standard way for the structure under consideration; then **Simulation** → **Run simulation**. Have in mind that a long computation time might happen as the matrix equation has to be set up and solved according to the chosen number of frequency samples.
4. Click on (tab **Postprocessing**) on investigate the computed system responses as a function of frequency. This is always recommended for validation. It is important that all system resonances are well approximated (always investigate the real part and the imaginary part). The displayed curves should look “smooth” and the value for the basic frequency should not deviate too much from the static value for f = 0, which cannot be computed numerically by *CONCEPT-II*.
5. Click on (time-domain tool) and select the type excitation function. Note that there is a clear relationship between frequency sampling (including the basic and the limiting frequency) and the possible parameters of the excitation function, for example the rise time, the total signal duration etc.. After the IFT has been performed the time-dependent curves appear in the

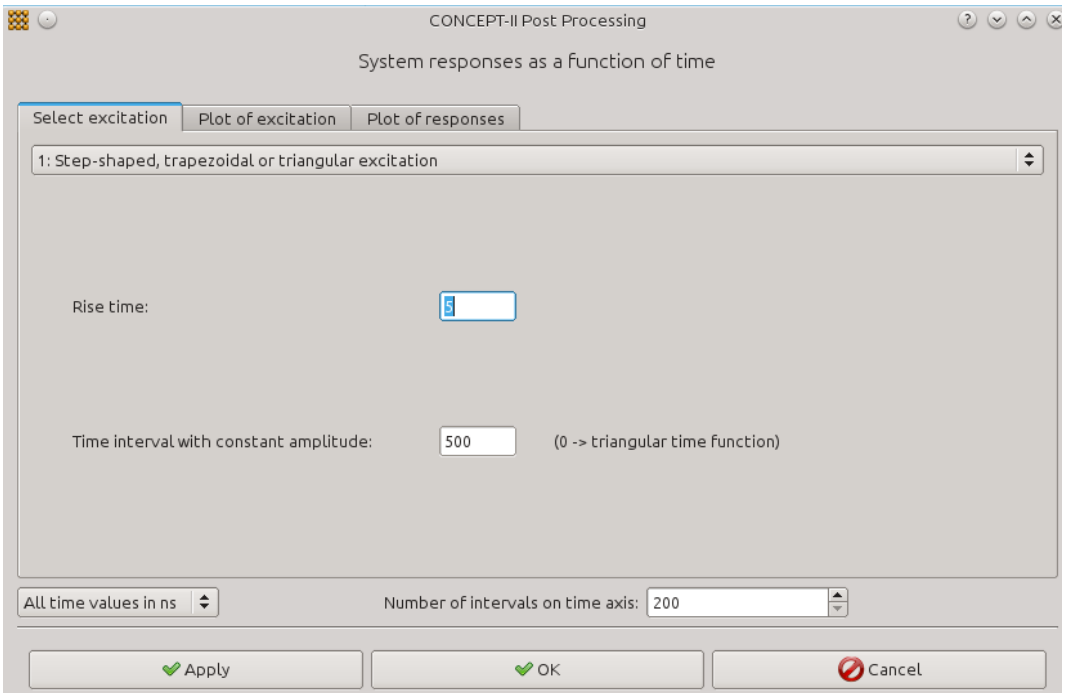


Fig 3.48: The Select excitation tab of the time-domain tool

Display area.

Currently the following excitation functions are available (top combo box in Fig. 3.48):

- Step-shaped, trapezoidal or triangular functions
- Double-exponential function
- Arbitrary time function. When chosen, a special window opens, where the number of time steps, the time steps and the corresponding function values can be entered. Hence the time function is approximated be piecewise linear sections
- sine-square pulse
- Lightning current function according to Heidler [Hei]

  Time values can be chosen in ns or in microseconds (in Fig 3.49 'All time values in ns' has been selected).

Under the tab **Plot of responses** (Fig. 3.49) is indicated which types of system

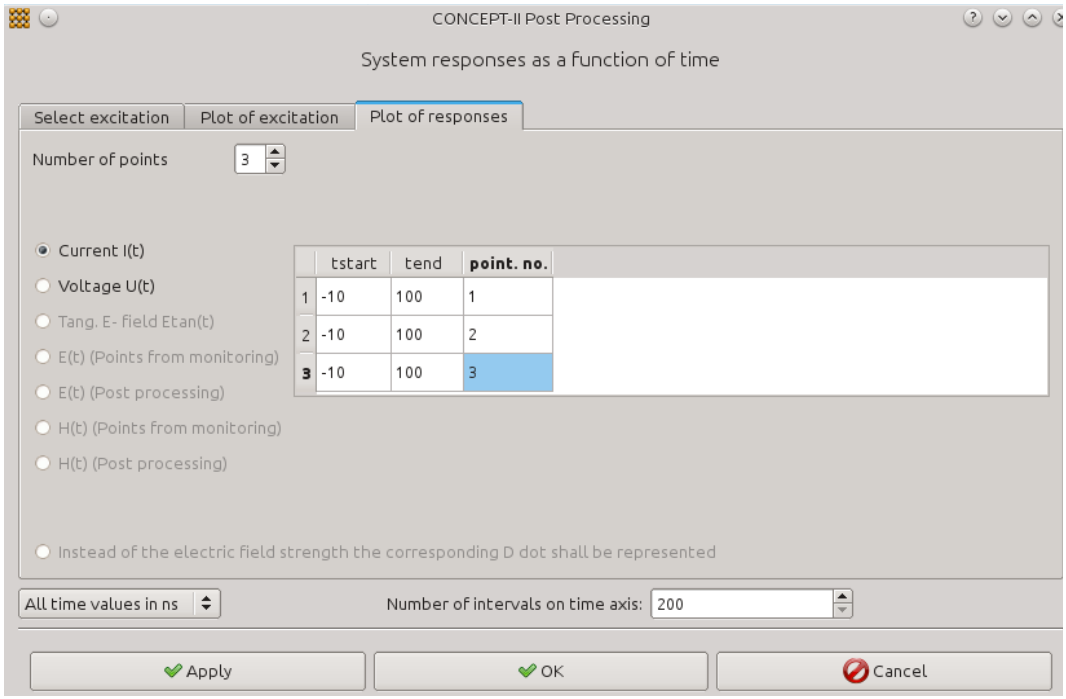


Fig 3.49: The **Plot of system responses** tab of the time domain tool

responses are ready for an inverse Fourier transform (IFT). In Fig. 3.49 *Current I(t)* Voltage *u(t)* are indicated and the current at observation points 1 2 3 are to be plotted in the time interval ranging from -10 ns to 100 ns.

Generally the system response which are subjected to an IFT should be specified in

the **Simulation** P*roject tree* (Simulation tab) already**,** at least voltage observation points and current observation points, also field points. Additional field points can be specified later by using (see **Post processing** tab)

Files: *time-e-n.asc, time-h-n.asc, time-u-n.asc, time-i-n.asc*, n of point (curve).

The example of a line excited by a voltage generator can be found under $CONCEPT/examples/Transient_line.

3.8.2 Key points for performing an IFT

The computation of time-domain system responses shall be demonstrated by means of an example which can be found under the directory *$CONCEPT/examples/Transient_line*

Problem description:
A wire loop above an ideal ground shall be investigated.

- Height of the wire loop: 0.1 m
- Length of the loop (horizontal wire x-directed): 5.0 m
- Radius of all wires: 1.75 mm

A voltage generator with zero internal impedance is located at the base point of the left vertical wire, corresponding to the origin of the rectangular coordinate system. At the opposite end, i.e. at the base of the right vertical section a resistor of 75 Ohm has been placed.
The structure can be seen in Fig. 3.50 .

Fig 3.50: Single-wire transmission line over an ideal ground excited by a voltage jump

Our goal is to compute the time dependence of the current at the beginning, the midpoint, and the end of the horizontal wire.

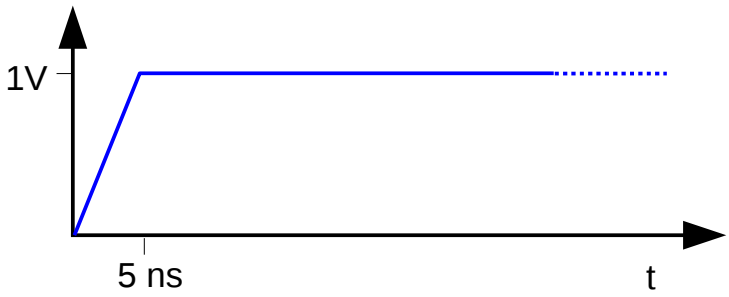


Fig 3.51: The excitation function

due to the excitation function provided by the generator (Fig. 3.51), which is a voltage step from 0 V to 1 V at a rise time of 5 ns.

General remarks

The user should note that “frequency sampling” means automatically that the corresponding time-domain responses will be repeated periodically although the generator in the physical structure provides a non-periodic excitation. In other words: a periodic time-domain function cannot be avoided due to the principles and rules of the inverse Fourier transform.

Furthermore it is important to note that a frequency-domain MoM code cannot consider extremely low frequencies; f = 0.0 Hz is not possible.

It follows that the total time duration of transient quantities to be computed is limited. The lowest frequency that *CONCEPT-II* can handle is in the range of 1 kHz for structures with typical dimensions of 1 ... 10 m. The total duration of transients should not exceed 0.25 ... 0.5 ms. Transient quantities, exciting functions and system responses must always start at zero amplitude and end with zero amplitude.

The computation of time-domain system responses caused by pulses with very short rise times can only be accomplished if sufficiently high frequency components are included. At these high frequencies the dimension of the system under consideration may comprise already many wavelengths leading to large equation systems and long computation times. Remember that the solve time for the equation system is proportional to N^3 with *N* as the number of unknowns and the LU decomposition is applied.

If currents tend to oscillate for a long time because energy can hardly be radiated or dissipated to heat in resistors it may get difficult to achieve satisfying results. Such systems, for example a single-wire transmission line close to the ground, terminated by a short-circuit or an open end, have sharp resonances. The “true” peaks of these resonances have to be captured resulting in a very short step width between the frequency samples. For each frequency step the system matrix has to be set up and the equation system has to be solved with the consequence of large computation times even in case of a moderate number of unknowns.

In *CONCEPT-II*, the frequency sampling is carried out according this scheme:

$f_0, 3 \cdot f_0, 5 \cdot f_0, 7 \cdot f_0, \cdots, f_g$ with f_0 as the basic frequency and f_g as the highest frequency.

In order to model a generator function as shown in Fig. 3.51 the transmission line has to be excited by a pulse as can be seen in Fig. 3.52.

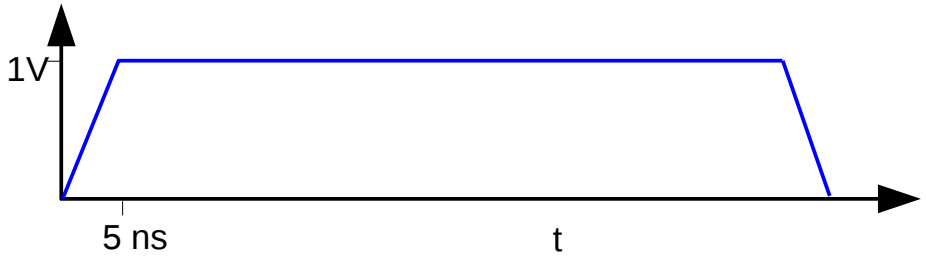


Fig 3.52: An excitation function must start with zero amplitude and end with zero amplitude

Important: Sampling in the frequency domain causes a periodic continuation of the signal in the time domain. Thus, due to the sampling according to mentioned scheme the excitation function which is **numerically** applied to the transmission line looks like the pink curve in Fig. 3.53. In this picture the basic frequency is $f_0 = 0.5\text{MHz}$ corresponding to a cycle duration of 2 µs, starting at T_0=0 and ending at T_p=2µs.

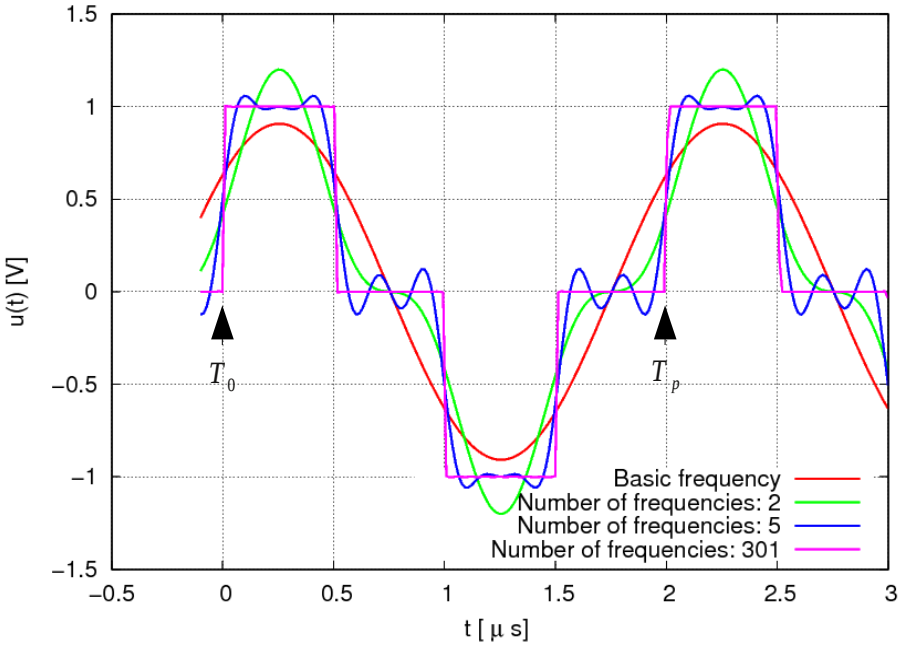


Fig 3.53: Forming of a time-domain signal u(t) by the superposition of sine functions. Pink curve: time dependence of generator voltage due to frequency sampling. The first frequency sample is f_0=0.5 MHz

We see that our transmission line is actually excited four times, namely at t_1=0 µs, t_2=0.5µs, t_3=1µs and at t_4=1.5 µs. Rise and fall times of the pink curve are 5 ns. Due to the frequency sampling according to $f_0, 3 \cdot f_0, 5 \cdot f_0, 7 \cdot f_0, \cdots, f_g$ the generator voltage is a time-

periodic bipolar signal.

At t=2 µs the voltage of the generator starts again. The transient voltages and currents which are caused by the last voltage jump from -1 V to 0 V at 1.5 µs should be damped to negligible amplitudes, otherwise aliasing will occur.

In order to get the **step response** of the system, the stationary state ("direct current") must be reached at t_1=0.5 µs, because at that instant of time the generator voltage begins to drop again.

From the overall length of the loop (5 m + 0.1 m + 0.1 m) we find the propagation time as follows:

t_{travel} = 5.2m/3.0e8 s = 5.2m/c= 17.33 ns .

The reflection factor is given by

$$r=\frac{R_2-\Gamma}{R_2+\Gamma}=\frac{(75\Omega-284.32\Omega)}{(75\Omega-284.32\Omega)}=-0.58$$ with R_2 as the terminating resistor and Γ as the characteristic impedance of the horizontal wire which can be calculated by the the well-known formula

$$\Gamma=60.0\cdot\ln(2h/r_w)\ ,$$

h: height of the horizontal wire,
r_w: radius of the horizontal wire.

From the magnitude of the reflection factor one can expect that current and voltage waves will travel back and forth for several times until the steady current of

I = 1 V/75 Ohm = 13.333 mA is reached.

We guess that 20 travel times are necessary to get the final current value. At t_1, t_2, t_3 quite similar transients will be excited resulting in

T_p = 4 * 20 * 17.33 ns = 1.36 µs (T_p: periodic time of the numerical exciting time function, we have chosen T_p as 2 µs, pink curve, basic frequency f_0, see Fig. 3.53)

Note: In the general case it may be difficult to predict T_p as one does not know the system behavior in advance. Low-loss system which easily tend to oscillate (example short-circuited line, open-ended line) require a very narrow frequency sampling. In such a case we will have sharp resonances in the frequency domain and long system responses in the time domain. As a rule of thumb one can say that transmission-line type structures (waveguide-typ structures) require a very fine frequency sampling if they are open-ended and/or short-circuited. In such cases energy can hardly be dissipated because radiation is negligible.

It is important to recognize that T_p includes the duration of the excitation function **and** the duration of the system responses, both have to reach their final values at T_p. In the case of

lightning induced electromagnetic quantities on vehicles (ships, aircrafts) the excitation function (impressed current) may have a dominant influence, i.e., traveling waves which might have been induced on the system are already damped to zero, while the exciting impressed current continues to flow.

$$T_p = 2 \cdot \text{total duration of excitation function}$$ (Lightning impact on vehicles)

On the other hand in the case of a sharp “needle-like” pulse excitation a structure will essentially be excited two times leading to $T_p = 2 \cdot \text{No. of travel times} \cdot \text{travel time}$

In general T_p depends on the following items:

- Typical geometrical extension of the configuration. Large structure cause long travel times
- Duration of the excitation function (may especially get dominant for lightning simulations)
- Type of the structure, antenna-type, transmission-line type. Lines can hardly radiate energy, the forming of resonance peaks are mainly caused by losses in the terminating resistors.

  Antennas radiate electromagnetic energy, therefore they not as critical as low-loss lines with respect to the frequency sampling. We usually get first resonances which are relatively broad although ideal conducting metal is assumed and no resistors exist
- Losses due to resistors or finite conductivity
- Other kinds of loads, for example inductors and/or capacitors, time constants L/R, R*C due to lumped loads.

In the example under discussion we have T_p = 1.36 microsecond which leads to the basic frequency of

$$f_0 = \frac{1}{T_p} = 0.74\,\text{MHz}$$

We have choose f_o = 0.5 MHz, allowing even more travel times along the loop. So frequency sampling starts at 0.5 MHz with a step width of 1 MHz.

Known from experience, good results in the time domain can be achieved by means of

$$T_{rise} \cdot f_g = 0.8 \cdots 1.5 \qquad (2)$$

with T_{rise} as the rise time in microseconds and f_g as the highest frequency in MHz.

From the last equation one finds

f_g = 1.5 / 0.005 = 300 MHz (chosen rise time is 5 ns)

Hence the number of frequency steps is around 300 MHz / 1 MHz = 300. The equation system has to be set up and solved 300 times by the *CONCEPT-II* back end.

Note: If high-quality system responses shall be achieved it is recommended to set the discretization of all wires according to the requirements of the highest frequency:

$$N_{wire} = \frac{L_{wire}}{\lambda_{min}} \cdot 8$$

with

N_w : number of unknowns (segments)
λ_{min} : wavelength at the highest frequency
8: minimum number of unknowns (basis functions) per wavelength.

The drawback of such a proceeding is a relatively high computation time if there are dominating wire parts in the structure under consideration.

If the propagation of current waves along long wires shall be displayed (, **Results Section**), the number of basis functions along that wire is not allowed to change.

Fig. 3.54 shows the frequency dependence of the computed wire current at the left vertical

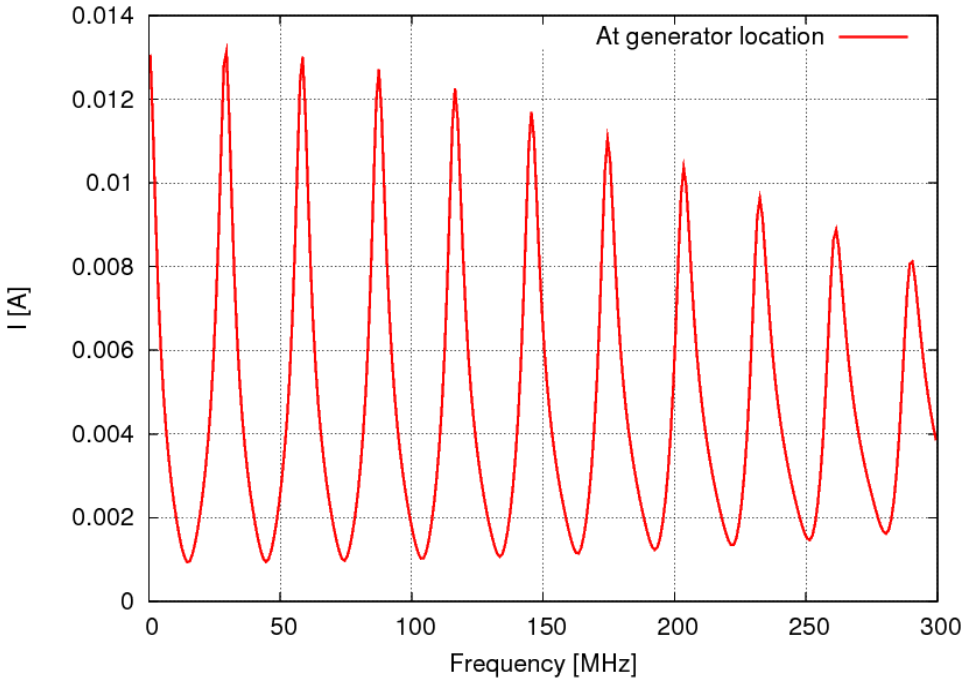


Fig 3.54: Magnitude of the wire current at the left vertical wire as a function of frequency

wire of the loop (, **Results section**). Two important features have to be emphasized,

which can be observed:

1. At f_0 (first frequency step) we have already a very good approximation of the static current which is 1V/75 Ohm = 13.3 mA. Therefore the time domain signal is supposed to have the right information once all propagation waves are damped to zero.

   General rule: f_0 should be such small that all physical quantities are already very close to their static values.

2. All resonance peaks are arranged “in a chain”.The envelope of all resonances appears as a smooth function which should always be the case. This indicates that the resonance peak values have been computed properly.

   General rule: Carefully observe if the true peak values of resonances are contained in the system responses.

   This requirement might be difficult to meet in the case of highly resonant structures.

It is always recommended to check the frequency domain system responses before performing an inverse Fourier transform. Time domain signals can be computed by means of time-domain tool: **Post processing** tab →

Mathematically we have to carry out an inverse Fourier transform of $Y(\omega)$

$$Y(\omega)=H(\omega)\cdot X(\omega)$$

with $H(\omega)$ as the transfer function computed by *CONCEPT-II*, based on an excitation with constant amplitude, and with $X(\omega)$ as the spectrum of the excitation function. By means of the **Select excitation** tab the user can specify various time-domain excitation functions (even arbitrary excitation functions) in order to perform the outlined operation.

In our example we select **Step-shaped trapezoidal of triangular function** with a *Rise time* of 5 ns and a *Time intervall with constant amplidude* of 500 ns. The time domain system response(s) shall be sampled by 200 time steps (*Number of intervals on time axis*). Especially complicated signals of long durations might require a much larger number of time steps.

Next the input positions under the **plot of responses** tab have to be completed (Fig. 3.55). We observe that data for the Current I(t) and Voltage U(t) and E(t) is available and activate the radio button for current plotting. The E(t) has the addition (*Post processing*), meaning the the observation points have been specified and computed under the Post processing tab with the fields tool and were not set in the front end (**Simulation** tab).

Here the currents at the beginning, middle and end of wire 2 (horizontal) shall be plotted,

each beginning at Tstart = -10ns and at Tend=250ns with a resolution of 200 time steps.

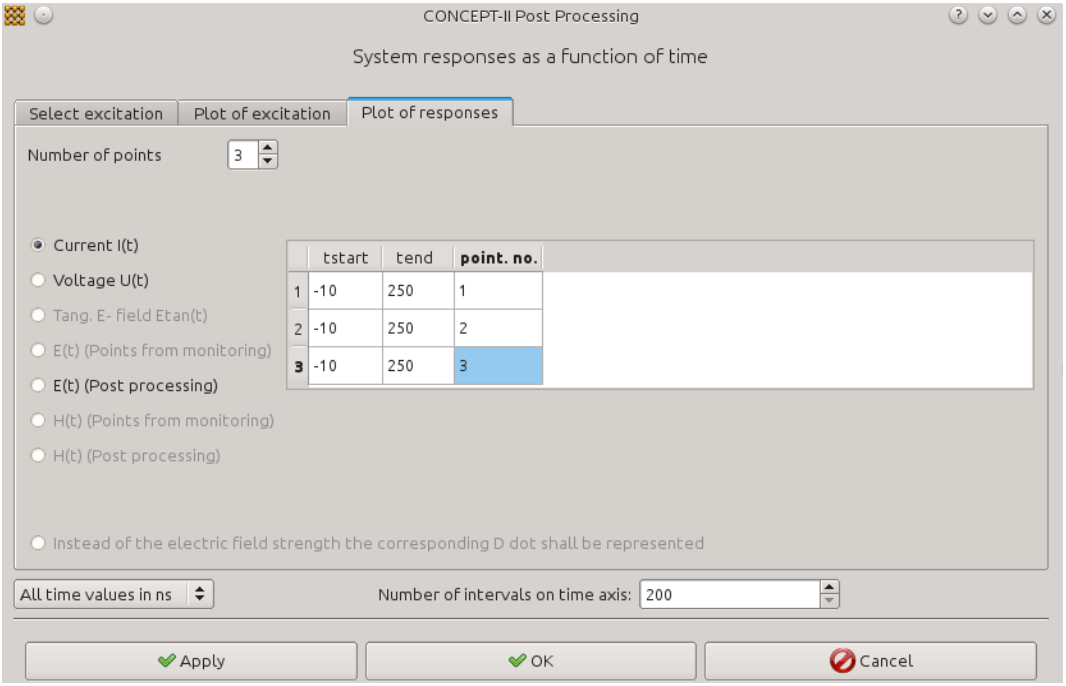


Fig 3.55: The 'Plot of responses' card of the inverse Fourier transform tool

Clicking on **Apply** provides the transient currents, see Fig. 3.56.

Useful hints: It is always a good idea to start the curve representations at neg. times, i.e. before the generator voltage begins to rise.

Here the time axis starts at -10 ns. Therefore one can see that the current at the generator position begins to rise exactly at 0 ns. Furthermore all depicted currents have zero magnitude before 0 ns. This would not be the case if the terminating resistor had a much smaller value or a much higher value (much higher than the characteristic impedance of the loop) and if the frequency sampling remained unchanged. Under such circumstances aliasing would occur. The user is invited to investigate this.

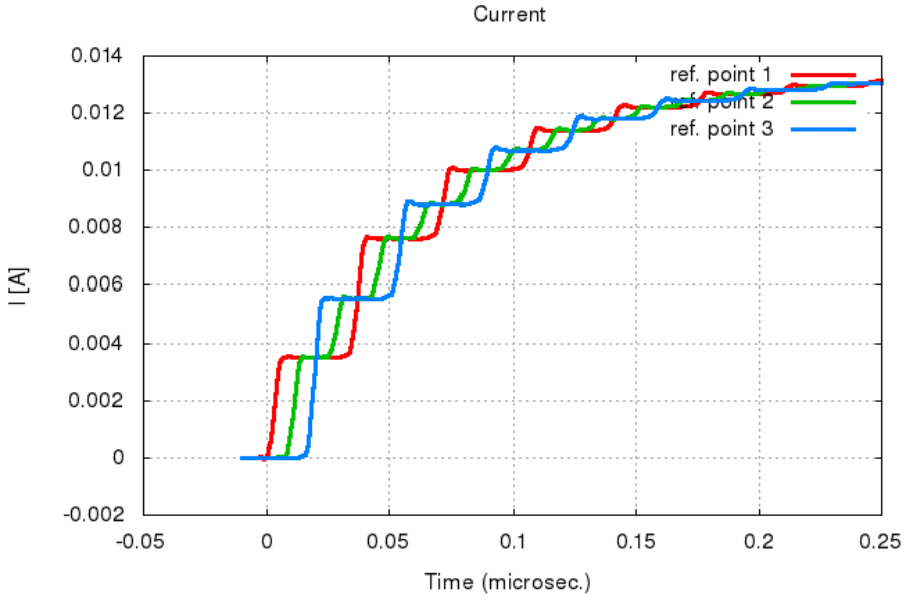


Fig 3.56: Computed wire currents as a function of time. Note the static values and time shifts of the functions according to the position of the observations points along the line.

A much shorter rise time would lead to oscillations with significant amplitudes especially before t=0. This is due to numerical reasons (bandwidth limitation) and is known as "Gibbs phenomenon". Some small amplitude oscillations can be observed in the current curves by looking more closely before t = 0.

Validating the results we find:

- All currents start from zero amplitude.
- All curves start according to the corresponding physical travel times.
- The final value of all current is identical. This is I = 1V/75 Ohm = 13.33 mA.
- The current at position 1 starts with 1 V/ Gamma = 1V / 284.32 Ohm = 3.52 mA

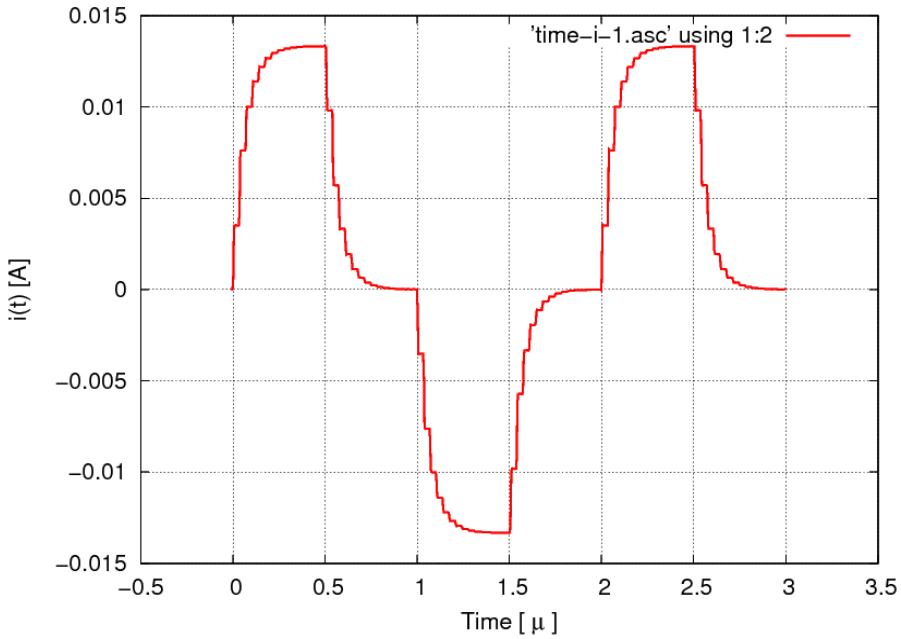


Fig 3.57: Computed wire current (left vertical wire) as a function of time, time interval extended

In Fig. 3.57 the time axis has been extended to 3µs. It can be observed that the wire loop is actually excited four times during T_p= 2µ according to theory. Each time the transient current decays completely before the next jump of the generator voltage happens.

Note:

- The time t = 0 corresponds to the feed point in the case of a generator excitation or the starting point of an impressed traveling current pulse.
- For a plane wave excitation t = 0 always refers to the origin of the coordinate system.
- Power input is not allowed if transient system responses shall be calculated.

3.8.3 Frequency sampling for an IFT

The frequency sampling for the structure under investigation can be specified in the entry fields of the window shown in Fig 3.47.

In the case according to Fig 3.47 each frequency step between step 1 and step 301 shall be considered by a full MoM computation, starting at 0.5 MHz (step width 1 MHz, according to the sampling scheme $f_0, 3\cdot f_0, 5\cdot f_0, 7\cdot f_0, \cdots, f_g$,see last section) and ending at step 301, corresponding to 300.5 MHz. “1” and “301” are called main step numbers or *Main frequency steps*. These numbers have to be entered, the corresponding frequency value will immediately be updated in the neighboring *Frequency* column.

If a *Step widths between main step numbers* were chosen to be 10 instead of 1, then the following sampling scheme between step 1 and step 301 would result: 1,11,21,31,...,301. Thus in such a case only every 10th step would be computed based an a full MoM analysis. The missing steps are automatically interpolated later when using and , respectively.

In case of a suitable behavior of the system responses as a function of frequency we could have an input situation as depicted in Fig. 3.58.

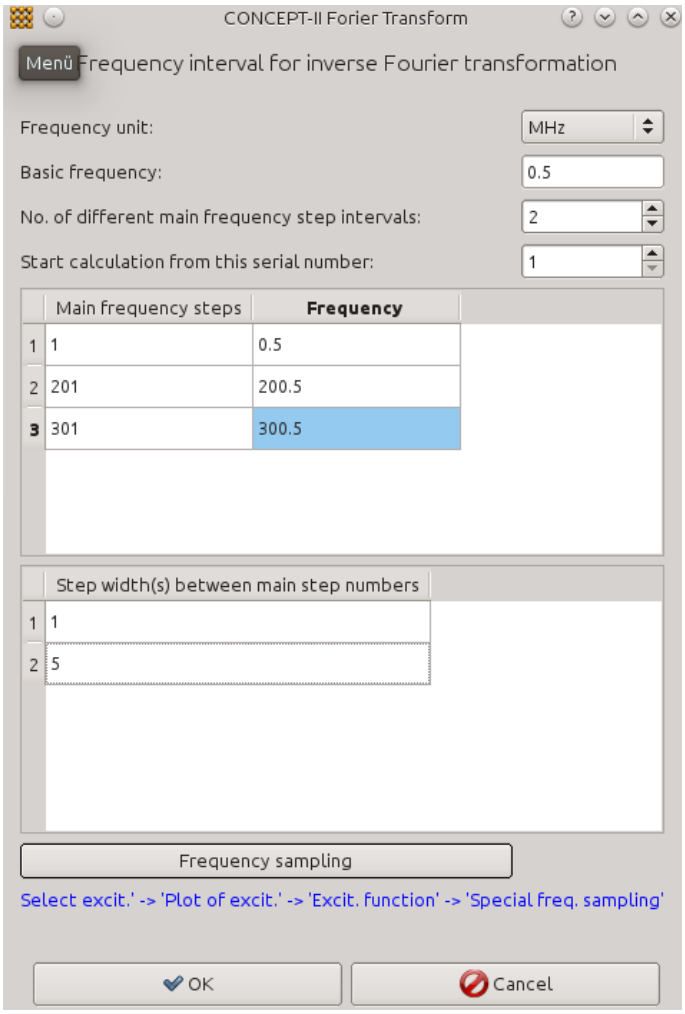


Fig 3.58: Frequency sampling for an IFT using different step widths

Now the interval between 0.5 MHz and 200.5 MHz is sampled with a step width of 2*0.5MHz = 1 MHZ. In the remaining region up to 300.5 MHz only every 5th step is explicitly taken into account, saving a lot of computation time.

If for example the *Start calculation from this serial number* were set to 2, the computation would start at step 206=201+5. Precondition would be that the results between the *Main frequency steps* 1 and 201 have already been computed successfully. Such a situation could happen for example for very time-consuming

problems where the first run revealed a rather smooth behavior of system responses, so that a rougher sampling seems to be justified for the next run. Another imaginable case: time domain quantities are insufficient due to an excessive appearance of Gibb's phenomenon, hence higher frequencies are to be included. So there is no need to start the whole computation starting at f_0 again.

Important: The difference of two successive main frequency steps must be divisible by the corresponding step width without remainder.

Clicking on **Frequency sampling** offers the possibility to investigate if the intended sampling scheme is sufficient for creating the excitation function before starting the back end run. Proceed as follows:

- The “System responses as a function of time” window opens. Click the **Select excitation** tab and choose the desired excitation function. Enter the required values (time constants, rise time etc.)
- Click the **Plot of excitation** tab
- Activate *Excitation function* and enter the start time and the end time for the curve representation into the corresponding input fields.
- Activate *Special frequency sampling* and enter the No. of frequency steps and the step width. If no values are entered the sampling scheme as specified under the *Freq. interval for inverse Fourier transform* (**Simulation** *Project tree*) is taken. Note that this can only be successful once the front end run has finished already! Only under this precondition the data is known to the tool.

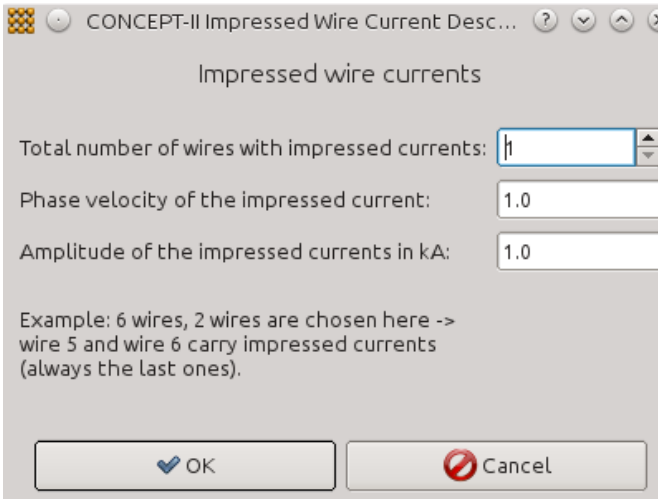
3.8.4 Special aspects: Impressed currents and lightning currents

Wire currents with known amplitudes can be taken as a structure excitation. Such currents which are developed in basis functions are related to the voltage vector only (right-hand side of the equation system, Sec. 4). Each basis function with known amplitude has impact on structure parts which are considered by the MoM equation system, in other words such a radiating basis function causes a voltage over each pair of neighboring patches and wire segments respectively - they induce currents. These induced structure currents have no impact back onto the impressed currents. Hence the known currents are 'impressed' ones.

A short section of a wire with a known constant current may be used as a current source. Spatially long wires (also electrically long wires) can be used to model the stroke of a lightning, either as a nearby stroke or as a direct stroke.

The electromagnetic fields of lightning return strokes can be modeled based on the transmission line model originally proposed be Uman [Uman]. The channel is taken into account by wires, which are carrying an impressed, i.e. known current distribution. Wires with impressed current distributions can be specified under the **Simulation** *Project tree*, item **Excitation → Impressed wire current**

The following window opens:


CONCEPT-II Impressed Wire Current Desc...
Impressed wire currents
Total number of wires with impressed currents:
1
Phase velocity of the impressed current:
1.0
Amplitude of the impressed currents in kA:
1.0
Example: 6 wires, 2 wires are chosen here ->
wire 5 and wire 6 carry impressed currents
(always the last ones).
OK
Cancel

- *Total number of wires with impressed current*

  It is important to have in mind that this number always refers to the last wire(s) which have been entered under the **Wires** (**Simulation** *Project tree*).

  Only a single wire file is possible in such a case.

- *Phase velocity of the impressed current*

  The propagating speed of the time-domain current function.

  0 → All basis functions have a constant amplitude all over the wire(s) with an impressed current distribution

  1 → Propagation with speed of light (0.333 is frequently taken for lightning simulations)

- *Amplitude of impressed current in kA*

  For lightning modeling purposes 0.001 kA is good choice in case of the function according to Heidler [Heidler]. The amplitude of the propagating exciting function can be specified later in the IFT tool which performs the inverse Fourier transform (**Post processing** tab)

  The lightning function according to Heidler: $$i = \frac{i_{max}}{\eta} \frac{\left(\frac{t}{T}\right)^{10}}{1+\left(\frac{t}{T}\right)^{10}} e^{\left(-\frac{t}{\tau}\right)}$$

  Parameters: Front time T, decay time τ, correction coefficient η, see input window of the IFT tool .

In case of a double-exponential function being used to simulate the event of a lightning the amplitude should be set in the front end run already. This value is related to the magnitude of the sinusoidal channel current distribution in the frequency domain. In the time domain we have this value in front of the exponential terms. This is not the true current peak amplitude of the time-domain function, which also depends on the chosen time constants. The time function that travels along the channel looks like this:

$$i(l,t) = i_{max} \cdot \left(e^{-\frac{1}{T_1} \cdot (t - l/v)} - e^{-\frac{1}{T_2} \cdot (t - l/v)} \right)$$

with l as the running coordinate along the channel and v as the speed of propagation.

- The time domain impulse propagates along the wire, starting at the beginning. If this point is attached to a wire or to the node of the surface patch of the affected structure this marks the point of strike. The type of excitation function can be chosen once the back end has finished ( )

- A curved channel in space can be modeled by straight sections where the end of each section is connected to the beginning of the next one. Branches are not allowed and only a single amplitude can be entered. In case of the channel being attached to the structure the continuity equation is fulfilled.
- Modeling of an ideal current source: A short piece of wire with an impressed current is attached to a structure being above a ground with ground connection or in the course of a closed loop.

  The presence of a completely closed current loop including the current return path is not necessary when using an impressed current distribution as excitation: the corresponding wire can stop or begin at a certain position in space.

  Simulating a lightning return stroke the channel length between the strike point and the cloud should be long enough to sufficiently include the effect of radiating EM fields (good results have been gained for channel length between 500m to 1km). This leads to many functions for a large channel lengths. Each current functions along the channel has an impact on all wire segments and surface patches of the MoM parts of the structure. Thus the computation of the right-hand side of the equation system gets time-consuming.
- Non-linear effect are not regarded in the computations.

According to Fig. 3.59 four different lightning functions are predefined for excitation. Clicking on **Sampling values** a suggestion for the frequency domain sampling is given under the **Plot of excitation** card. Clicking on **Apply** provides the corresponding curve on the screen. Note that *Step width* corresponds to $2*f_0$.

A good value for the number of time steps (*Number of intervals on the time axis*) for the time-domain curve representation is 200 (default). For complicated signals with superimposed oscillations and a long duration 600...800 intervals (steps) are suggested.

An example for aligning simulation can be found under *$CONCEPT/examples/Lightning_overhead_line*

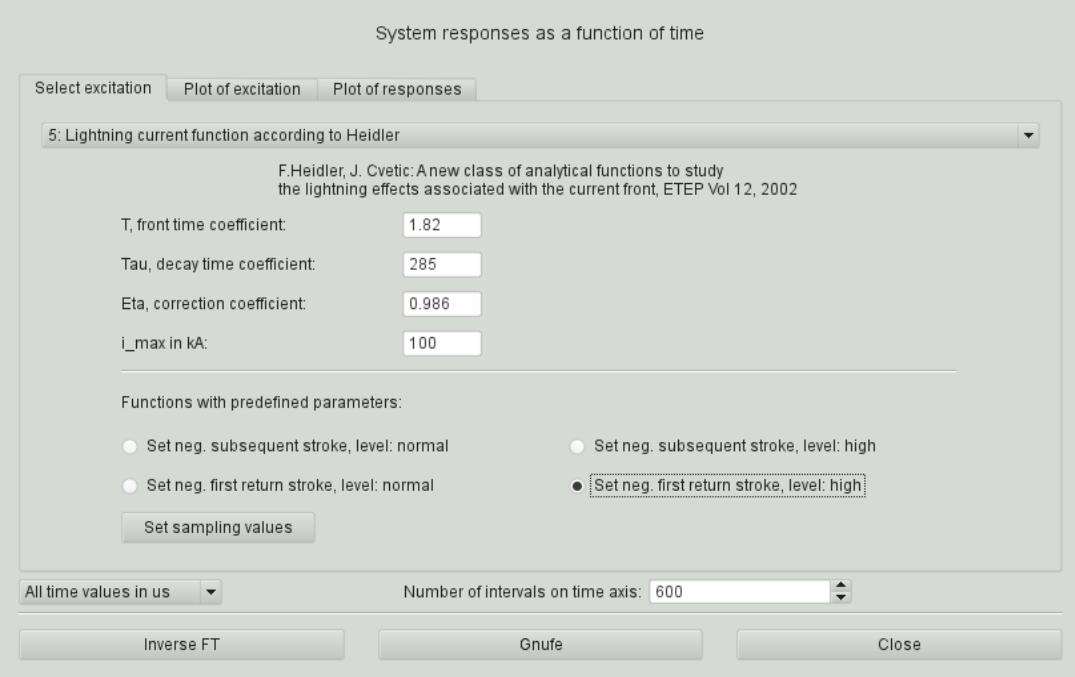

System responses as a function of time
Select excitation
Plot of excitation
Plot of responses
5: Lightning current function according to Heidler
F.Heidler, J. Cvetic: A new class of analytical functions to study
the lightning effects associated with the current front, ETEP Vol 12, 2002
T, front time coefficient:
1.82
Tau, decay time coefficient:
285
Eta, correction coefficient:
0.986
i_max in kA:
100
Functions with predefined parameters:
Set neg. subsequent stroke, level: normal
Set neg. subsequent stroke, level: high
Set neg. first return stroke, level: normal
Set neg. first return stroke, level: high
Set sampling values
All time values in us
Number of intervals on time axis:
600
Inverse FT
Gnufe
Close

Fig 3.59: Input parameters according the lightning current defined in the international standard IEC 61312-1

[Uman] M.A. Uman, D.K. McLain, “Magnetic Field of Lightning retirn Stroke”, Journal of Geophysical Resaerch, vol. 74, Dec. 1969, pp. 6899-6910

[Heidler] F. Heidler, J. Cvetic, “A Class of Analytical Functions to Study the leightning Effects Associated With the Current Front, ETEP Vol. 12 March/April 2002, pp 141-149

3.9 Using CONCEPT-II for parallel computation

3.9.1 Linux

All machines forming a cluster (distributed memory systems) should have the same operating system and the same size of memory. The cluster can be formed for example by either a single shared memory machine or by several shared memory machines.

Note that for configuring a cluster a certain knowledge of the Linux operating system is required! **The cluster will not be configured by the** *CONCEPT-II* **team** (there are many ways to do this). It is only described what requirements are necessary to successfully run the package on a cluster.

Each of the participating computers must be able to connect to any other machine without being asked for the password (this is a question of configuring `ssh` in an appropriate manner). Normally, if `ssh` is used to connect to a remote host, this host will prompt for the password. There is need to be able to execute jobs on remote nodes without typing in a password.

Format of the machine file: (necessary on distributed memory systems)

Node name 1 slots = **Number of CPU's** on the computer

Node name 2 slots = **Number of CPU's** on the computer ...

Determine the problem size: $N \cdot N \cdot 16/1024^3$ GByte are needed (double precision), where *N* is the number of unknowns. **Note that parallel out-of-core solutions are not supported by the** *CONCEPT-II* **parallel back-end.**

Commands at the prompt for distributed PCs:

mpirun -np n –machinefile {name of machine file} concept.be

Command on a single multicore PC:

mpirun -np n concept.be

“n” is the number of processors that are available

Use *concept.sbe* for computations in single precision.

3.9.2 Windows

Currently the parallel solvers have only been tested on shared memory machines.

For further details see the installation instructions.

3.10 Format of surface patch files and wire files

Surface patch files have the following format (ASCII data):

Line 1: Number of nodes N_{node} , number of surface patches $N_{element}$

Line 2, coordinates of node 1: x y z

Line 3, coordinates of node 2: x y z

...

Line $N_{node}+1$ x y z

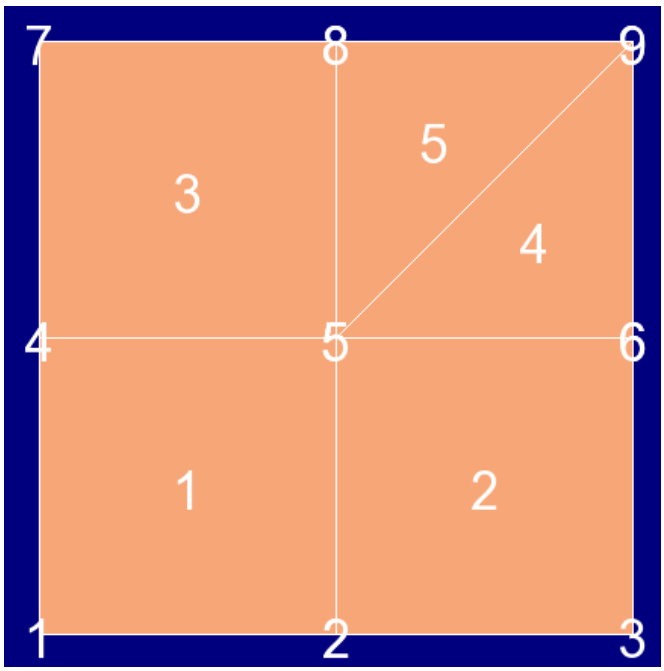
Line $N_{node}+2$ node numbers of patch 1

Line $N_{node}+3$ node numbers of patch 2

...

Line $N_{node}+1+N_{element}$ node numbers of the last patch

Example: square plate in the x-y plane, edge length: 1 m.



Contents of the surface patch file:

```
 9  5                          (9 nodes, 5 patches)
  0.0  0.0  0.0    ( first node)
  0.5  0.0  0.0    (second node)
  1.0  0.0 0.0     (third node)
  0.0  0.5  0.0
  0.5  0.5  0.0
  1.0  0.5  0.0
  0.0  1.0  0.0
  0.5  1.0  0.0
  1.0  1.0  0.0     (node number 9)
  1  2  5  4   (description of the first patch, see above figure, quadrangle, 4 nodes)
  2  3  6  5  (description of the second patch, quadrangle, 4 nodes)
  4  5  8  7
  5  6  9  0 (triangle, last node number is a “0”, patch number 4)
```

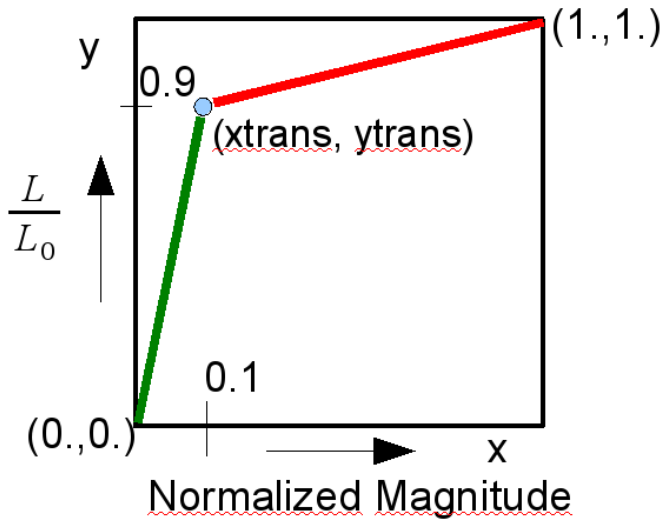
5 9 8 0 (triangle, patch number 5, also triangle)

The wire format looks like this (R. Radius, NB: number of basis functions):

1st line: number of wires N
2nd line Xstart Ystart Zstart Xend Yend Zend R NB→ 1st wire
3rd line Xstart Ystart Zstart Xend Yend Zend R NB→ 2nd wire
4th line Xstart Ystart Zstart Xend Yend Zend R NB
...
N+1th line Xstart Ystart Zstart Xend Yend Zend R NB→ coordinates Nth wire

3.11 Arrow scaling, xtrans, ytrans

For the representation of vector diagrams the arrows have to be scaled in an appropriate way. In practice it is sometimes advantageous to display the vector in regions of very small amplitudes in a linear scale. This can be carried out with respect to the surface patch current distribution ( ) or 2D representations of the electric field strength, see for example. The scaling method is based on a unit square defined in an x-y coordinate system as shown in the next figure.



The standard values of xtrans, ytrans are (0.1, 0.9), see above figure. This means that 90% of the default maximum arrow length L_0 is linearly mapped to all values

of the depicted quantity between 0 and 10% of maximum value of this quantity. These arrows are shown in green color. All remaining arrows are allowed to vary between 90% and 100% of L_0 and are shown in red. L_0 depends on the grid size and the average surface patch size, respectively. For a good representation it is recommended not to have more then 10% of all arrows in red.

Setting xtrans to a lower value, for example 0.01, means that regions with very small current or field vectors are magnified, so that their behavior can be observed . As no log. scale is applied one gets a good “feeling” of the physical situation.

Setting both xtrans and ytrans to 1 causes a linear scaling of all arrows – no red arrows will appear.

Setting ytrans to 1 (and leaving xtrans for example at 0.1) causes all arrows larger than 10% of the maximum field or current vector to be drawn in yellow. All yellow arrows have the default maximum arrow length L_0 .

3.12 Backup of project directories

Not all files need to be saved. Several levels can be distinguished:

1. Backup with the lowest storage requirements: Save *project.sim* and all wire and surface files that are named in *project.sim* and *confreq.in*

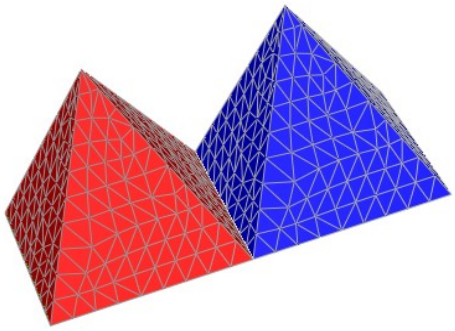
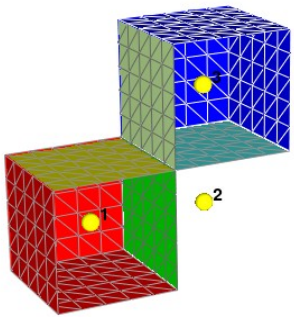
2. Add computed currents and fields: *co_ifl(*n*).bin, co_ili(*n*).bin.* In order to be able to use this data, it is necessary to perform the following steps: Load the simulation, then click on the **Simulation** button in the Menu bar → **Run Simulation front end**. The Post processing tools are ready to be used now.

3. Save the whole project directory.

3.13 Known problems

Here is a list of currently known bugs and problems (Feb. 2014):

1. Only Windows: If a parallel computation has been killed the files are not free for a few minutes. Hence do not restart the simulation until the files are free again.

2.  

   Structures as shown in the pictures above (for example 2 pyramids of dielectric material on ideal conducting metal plates or 2 dielectric bodies being attached at a common boundary)

3. Proper setting of the clipping plane in the *Display area*.
4. Problem under Windows operating systems when enlarging a part of the structure by dragging the mouse pointer. The corresponding area becomes non-transparent while selecting (Qt-problem).
5. Opacity might lead to a faulty representation of background surface parts
6. Small gaps which are sometimes hardly detectable by global inspection of the grid structure of body-related surfaces might abort the front end already. An error message will not be provided. Workaround: **Cad Tools** → **Geometry from simulation** → **Load simulation files** → **Cad tools 2** → → activate *Find free nodes* → **Apply**

   The file *wire.freenodes* is generated and shows all free edges. Erroneous regions (small gaps can be detected). Eventually set all surface patch files to invisible or enlarge the line width for the wires for better visibility.

4 Theoretical background (method of moments, EFIE, MFIE)

Basics of the method of moments

The method of moments (MoM) has become an important numerical method and is firmly established in the range of applied electromagnetics. It is a general procedure for converting an operator equation into a linear equation system which can be solved on a computer. Its name evolves from the process of taking moments by multiplying by appropriate weighting functions and integrating. The use in electromagnetic problems has become popular especially since the work of R. F. Harrington [Harr1, Harr2]. Recommended for further reading is also [Gibson].

When applying a computer code, the user should have certain knowledge of the

mathematical background. For this reason the main principles are shortly repeated here. Following Harrington consider a homogeneous equation of the form

$$L(f)=g \tag{2}$$

where L is a linear operator (for example the electric field integral equation, EFIE), g is a known function. This can be for example an impressed plane wave field, illuminating the structure. *CONCEPT-II* only considers cases where L represents integro-differential equations. The function f is the response of the excitation and is always represented by pure electric currents, i.e. surface currents and/or wire currents. Both f and g are functions of spatial coordinates and frequency; we seek to determine u with both g and L known.

The unknown function f is represented by a series expansion in the domain of L as

$$f=\sum_{(n=1)}^{\infty}\alpha_n f_n \tag{3}$$

where the α_n are constant coefficients which are to be determined. The functions f_n are called basis functions. Substituting (2) into(3)and assuming linearity gives

$$f=\sum_{(n=1)}^{N}\alpha_n(Lf_n)\simeq g \tag{4}$$

A set of testing, functions w_m is defined and the inner product, taken to be the integral of the product of functions is formed according to

$$\sum_{(n=1)}^{N}\alpha_n\langle(Lf_n,w_m)\rangle\simeq\langle g,w_m\rangle \tag{5}$$

Eq. (5) represents a set of N equations and can be written in matrix form as

$$[\tilde{L}]=\begin{bmatrix}\langle L_e(f_1),w_1\rangle & \langle L_e(f_2),w_1\rangle & \cdots\\ \langle L_e(f_1),w_2\rangle & \langle L_e(f_2),w_2\rangle & \cdots\\ \cdots & & \\ \langle L_e(f_1),w_N\rangle & \langle L_e(f_2),w_N\rangle & \cdots\end{bmatrix},\ [\vec{\alpha}]=\begin{bmatrix}\alpha_1\\ \alpha_2\\ \vdots\\ \alpha_N\end{bmatrix},\ [\vec{g}]=\begin{bmatrix}\langle g_1,w_1\rangle\\ \langle g_2,w_2\rangle\\ \vdots\\ \langle g_N,w_N\rangle\end{bmatrix},$$

$$[\tilde{L}][\vec{\alpha}]=[\vec{g}] \tag{6}$$

with $[\tilde{L}]$ is a N x N matrix.

Practically the solution for the α_n is gained by LU decomposition and according to (3) an approximation of the function f is known.

Thus, the general procedure starts with the formulation of an integral equation of the physical problem, i.e. we have to find an appropriate integral equation representing the operator L . By appropriate choices of basis functions functions

and testing functions, the integral equation is turned into a matrix equation to be solved.

Integral equations for wires and metallic surfaces

For wires the electric field integral equation looks like this:

$$\vec{E}^{inc}(l)\cdot\vec{e}_l = \int_0^L \left[\frac{j}{\omega\epsilon_0} \frac{dI(l')}{dl'} \frac{\partial G(\vec{r},\vec{r}')}{\partial l} + j\omega\mu_0 I(l') G(\vec{r},\vec{r}') \vec{e}_l \cdot \vec{e}_l' \right] dl' \tag{7}$$

$\vec{E}^{inc}(l)\cdot\vec{e}_l$: Tangential impressed (known) field along a wire with running coordinate *l*.

G : Green's function of free space

The following electric field integral equation (EFIE) is applicable to general metallic surfaces with ideal conductivity (e.g. plates, shells, closed cavities or cavities with apertures):

$$\vec{n}\times\vec{E}^{inc} = \vec{n}\times\left\{ \frac{1}{4\pi} \iint_S \left[j\omega\mu\vec{J}_s \frac{e^{-jkR}}{R} + \frac{j\,\mathrm{div}'\vec{J}_s}{\omega\epsilon_0} \mathrm{grad}\frac{e^{-jkR}}{R} \right] dS' \right\} \tag{8}$$

$$\vec{n}\times\vec{E}^{inc} = \vec{n}\times\vec{E}(\vec{J}_S)$$

$\vec{J}_s$: Surface current distribution

$\vec{n}$: Normal vector of the surface

Using line matching with suitable testing functions (matching of the boundary condition $E_{\tan}=0$) these integral equations can be converted into

$$[Z]\cdot[I]=[U] \tag{9}$$

With $[Z]$ as the impedance matrix, $[I]$ as the vector with the unknown complex current amplitudes and $[U]$ as the voltage vector with all its elements known, compare to (6).

One major limitation of the method is the large matrix size needed to solve a problem accurately if the geometry is large and complicated and/or high frequencies are to be investigated. In these cases, the typical structure dimensions are much larger than the wavelength.

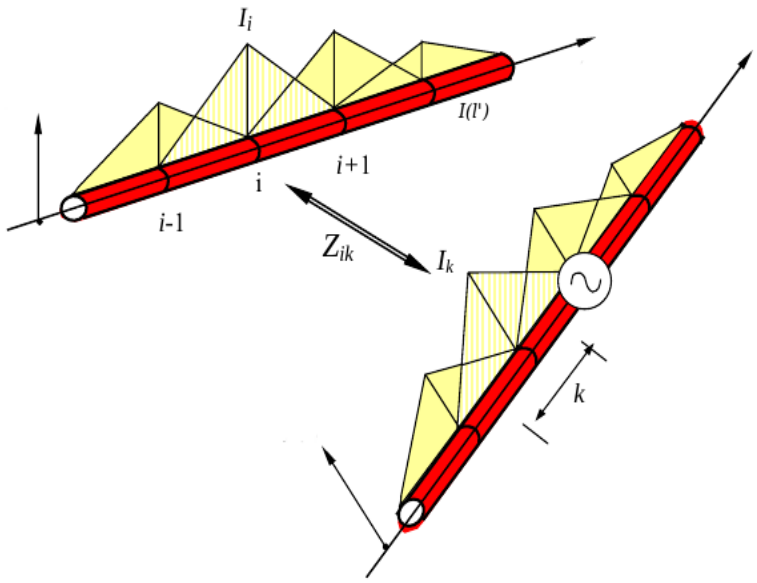
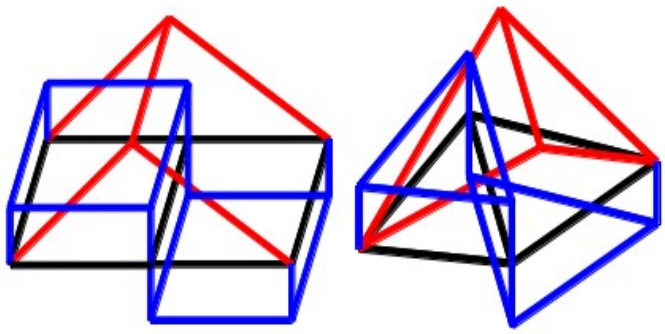


Fig 4.1: Triangular basis functions along two wires arranged close to each other

For wire structures we only have to perform an integration over line sources (current filaments along the axis). In *CONCEPT-II* triangular basis functions are applied as illustrated in Fig 4.1. From the overlapping of neighboring triangles results a piecewise linear approximation of the current. The distribution of the line charges along a wire is approximated by a step function.

Fig 4.2: Roof-top funtions (left) and RWG functions (right)



Setting up the system matrix can be explained as follows: Z_{ik} is the complex impedance for the impact of the basis function with amplitude I_i on wire segment k, causing a voltage U_k over segment k (without the corresponding wire being physical present). Then the matching position k+1 is considered and again the

effect of all current basis functions is computed in an

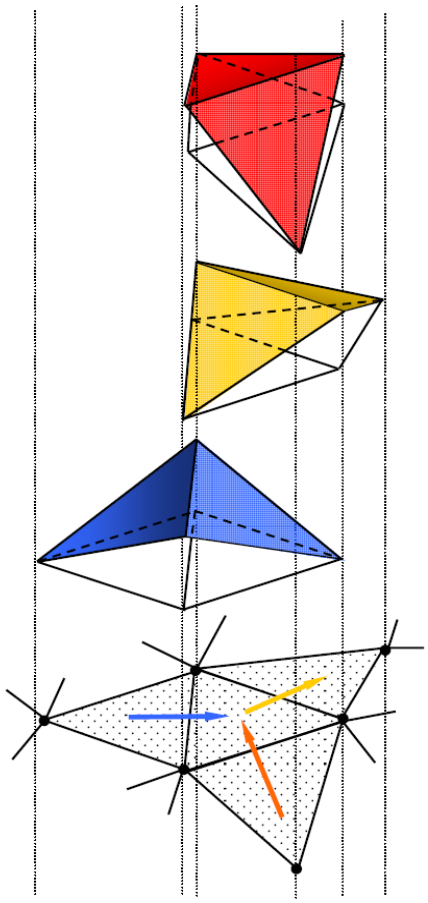


Fig 4.3: Superposition of RWG basis functions in a triangle

analogue manner.

Finally we get an equation system with as many lines as currents being defined on the wires (voltages are determined “under the tip of each triangle”).

Due to assumtion of infinite conductivity, the line integral of the tangential E field along the wires vanishes except for one segment, where the voltage generator has been defined. Therefore the voltage vector in (9) contains only zero coefficients except for a single entry, corresponding to the location of the generator.

On metallic surfaces, so-called RWG functions [RWG] defined over adjacent triangles and roof top functions defined over adjacent quadrangles are applied. Triangles can be attached to quadrangles. Fig 4.2 illustrates the surface basis functions: Black lines designate the patch boundaries, the magnitude of the current distribution on the patches is marked in red lines and finally blue lines indicate the surface charge distribution related to the falling and rising current parts. Note that the surface current density vectors are oriented from the left patch to the right patch in each case, they are so-called vector basis functions. Setting up the system

matrix Z works quite similar to the approach already explained for wires, again the boundary condition that the tangential E field vanishes along ideally conductors is applied. Once the equation system is solved the complex amplitudes of the the vector basis functions are known and further quantities like EM fields in arbitrary observation points, along spatial paths or in regions in space can be computed by integrating over all patches (and wires segments, is wires are present).

In Fig 4.3 is illustrates the superposition of the RWG basis functions on a patch; at each edge another patch is connected. Knowing the amplitudes of the blue, yellow and red basis functions the resulting current pattern on the center patch can be computed. In the centroid we will have a certain magnitude and direction of the current density vector. The tool works in the described way, displaying the overall surface current distribution in the *Display area*.

Treatment of dielectric bodies

In the case of a dielectric body we have to consider 2 boundary conditions:

$E_{t1}=E_{t2}$ $H_{t1}=H_{t2}$ We have a set of two integral equations to be solved:

$$\vec{n}\times\vec{H}_1(\vec{J}_1)=\vec{n}\times\vec{H}_2(\vec{J}_2)$$
$$\vec{n}\times\vec{E}_1(\vec{J}_1)=\vec{n}\times\vec{E}_2(\vec{J}_2)$$

Introducing the corresponding operators we have:

$$\vec{n}\times\left\{\frac{1}{4\pi}\iint_S \vec{J}_1\times\mathrm{grad}\frac{\mathrm{e}^{-jk_1R}}{R}dS'+\vec{H}^{inc}\right\}=\vec{n}\times\left\{\frac{1}{4\pi}\iint_S \vec{J}_2\times\mathrm{grad}\frac{\mathrm{e}^{-jk_2R}}{R}dS'\right\}$$ and

$$\vec{n}\times\left\{\frac{1}{4\pi}\iint_S\left[j\omega\mu_1\vec{J}_1\frac{\mathrm{e}^{-jk_{1R}}}{R}+\frac{j\,\mathrm{div}'\vec{J}_1}{\omega\epsilon_1}\mathrm{grad}\frac{\mathrm{e}^{-jk_{1R}}}{R}\right]dS'\right\}=$$
$$\vec{n}\times\left\{\frac{1}{4\pi}\iint_S\left[j\omega\mu_2\vec{J}_2\frac{\mathrm{e}^{-jk_2R}}{R}+\frac{j\,\mathrm{div}'\vec{J}_2}{\omega\epsilon_0}\mathrm{grad}\frac{\mathrm{e}^{-jk_{2R}}}{R}\right]dS'\right\}$$

.

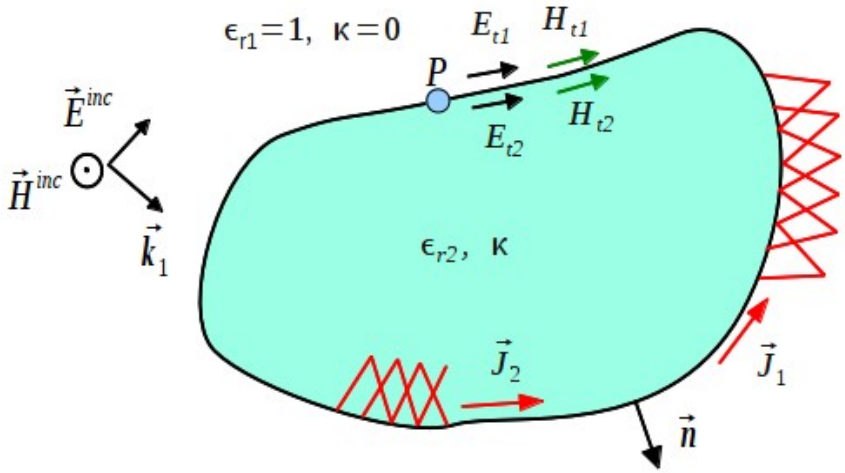


Fig 4.4: A dielectric body illuminated by a plane wave field

As is illustrated in Fig 4.4 we now have two current distributions, $\vec{J}_1$ is causes the scattered field outside the body and $\vec{J}_2$ causes the internal field. Both currents are equivalent current distributions the purpose of which is to ensure continuous tangential field components at the boundary of the body. According to the above explained MoM approach an equation system has to be set up – compared to the case of metallic body with different boundary conditions and with twice the number of unknowns.A comprehensive study of boundary integral equations can be found in [Harr3]

Thin layers on dielectric bodies

Fig 4.5 depicts a completely closed cavity. The walls of the cavity have a finite conductivity perhaps in conjunction with a certain relative permeability and are very thin with the thickness w. It is assumed that the typical dimension of the cavity D is much larger than w. In practical cases D might be in the m range whereas w is only a fraction of a mm, or even smaller. Such structures can hardly be solved by numerical methods from various reasons. In *CONCEPT-II* a hybrid technique has been implemented to overcome these problems.

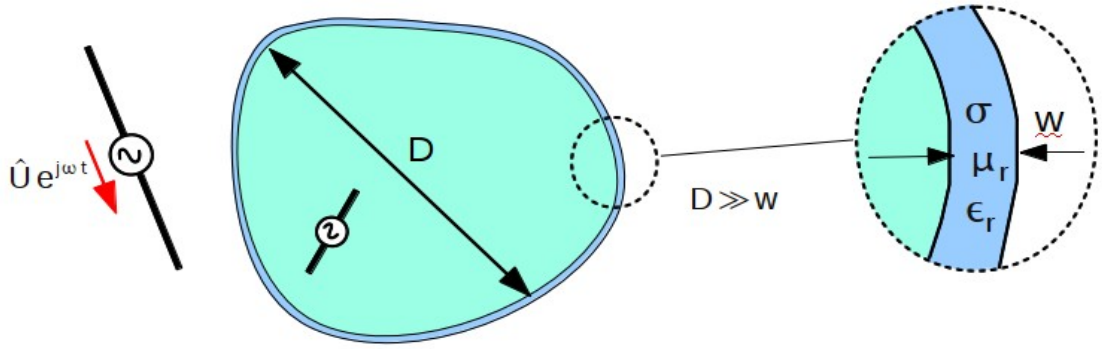


Fig 4.5: A shielded enclosure excited by a monopoe antenna

As shown in Fig 4.5 an antenna is illuminating a cavity causing induced currents on the metallic surface. Due to the finite conductivity and small thickness w of the wall, the external fields are able to penetrate into the interior. Inside the wall the electromagnetic fields are oriented in parallel to the wall boundaries and will propagate almost perpendicular to the walls, i.e., locally we will have a wave propagation that can be described by the following equations.

$$\vec{E} = \vec{A}\, e^{j\vec{k}\cdot\vec{\xi}} + \vec{B}\, e^{-j\vec{k}\cdot\vec{\xi}}$$

$$\vec{H} = -\frac{\vec{k}}{\omega\mu} \times \left(\vec{A}\, e^{j\vec{k}\cdot\vec{\xi}} - \vec{B} e^{-j\vec{k}\cdot\vec{\xi}} \right)$$

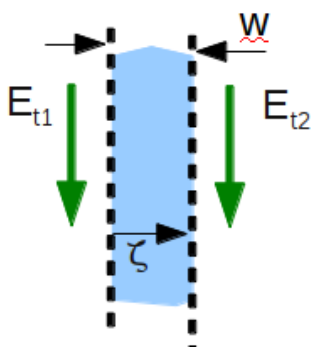


Fig 4.6: Local situation in a cross-section of the cavity walls, compare with the indicated position in Fig 4.5

The external tangential fields (for example the components E_{t1}, E_{t2} as shown in Fig 4.6) are caused by the currents induced on the external and internal sides of the cavity. Requiring that the EM fields are continuous on both sides of the cavity walls the constants A and B in the wave equations can be eliminated.In other words: the MoM system of equations is set up in a standard manner for a dielectric body, assuming air ($\epsilon_r = 1$) in this special case, see Fig 4.7. Instead of enforcing the boundary conditions directly the given wave equations are “switched in between”. Developing the theory further is turns out that the coefficients of the original equations system have to be weighted and rearranged to be able to compute the current distributions which are responsible for the EM fields inside and

outside the cavity.

Note that the wall thickness and the wall material is contained only in the analytical formulas for the internal wave propagation, hence the fields inside the wall are not resolved explicitly. Note further that the condition D >> w has to be fulfilled. In practice the approach finds its limits when w gets larger then 5...6.times the penetration depth.

Although the cavity according to Fehler: Referenz nicht gefunden has a finite thickness w it is modeled by single surface patch file, which has to be discretized in the standard way, carrying two electric current distributions. The user can imagine the surface being located at the center of the wall. This approach becomes critical, of course, if w becomes too large compared with D.

It is no problem to consider multilayer sheets, however, we have the same restrictions for the electrical and geometrical thickness imposed on the resulting wall thickness as mentioned before for a single layer.

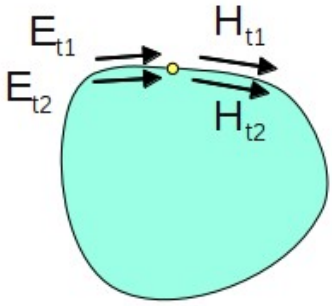


Fig 4.7: The boundary conditions to be fulfilled an a standard dielectric body.

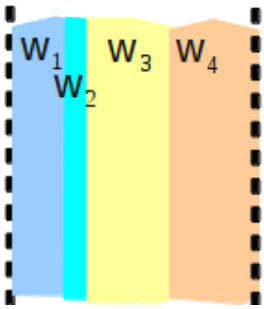


Fig 4.8: A multilayer sheet.

[Harr1] R. F. Harrington, Field Computation by Moment Methods, Macmillan Co., New York, 1968.

[Harr2] R.F. Harrington: Matrix Methods for Field Problems, Proceedings of the IEEE, vol. 55, no. 2, February, 1967

[Gibson] Walton C. Gibson: The method of Moments in Electromagnetics, Capman & Hall/CRC, 2008

[RWG] S. M. Rao, D. R. Wilton, A. W. Glisson: Electromagnetic Scattering by Surfaces of Arbitrary Shape. IEEE Trans. Antennas and Propagation, vol. AP 30, no. 3, May 1982, pp. 409-418

[Harr3] R.F. Harrington: Boundary Integral Formulations for Homogeneous Bodies, Journal of Electromagnetic Waves and Applications, vol. 3, number 1, 1989 , pp. 1-15(15)